Oasys D3PLOT
Version 9.3

Oasys

The Software house of Arup

For help and support from OASYS please contact:

UK

Arup Group Ltd
The Arup Campus
Blythe Gate
Blythe Valley Park
Solihull
West Midlands
B90 8AE
United Kingdom
Tel: +44 (0) 121 213 3399
Fax: +44 (0) 121 213 3302
Email: dyna.support@arup.com
Web: www.oasys-software.com/dyna

China

Arup
39/F-41/F
Huai Hai Plaza
Huai Hai Road (M)
Shanghai
China 200031
Tel: +86 21 6126 2875
Fax: +86 21 6126 2882
Email: china.support@arup.com
Web: www.oasys-software.com/dyna

India

Arup
Plot 39, Ananth Info Park
Opp. Oracle Campus
HiTec City
Madhapur Phase II
Hyderabad 500081
India
Tel: +91 40 4436 9797/98
Email: india.support@arup.com
Web: www.oasys-software.com/dyna

or contact your local Oasys distributor

LS-DYNA, LS-OPT and LS-PrePost are registered trademarks of Livermore Software Technology Corporation

0 Preamble **0.1**
Acknowledgements 0.1
Abstract 0.1
Host computers 0.1
Memory requirements 0.1
Output devices 0.1
New Features for Version 9.3RC1 0.2
New Features for Version 9.2 0.2
New Features for Version 9.0 0.3
New Features for Version 8.3 0.3
New Features for Version 8.2 0.4
New Features for version 8.1 0.4
New Features for version 8.0a 0.4
Manual Revision History 0.5
Text conventions used in this manual 0.5
1 LS-DYNA FEATURES SUPPORTED BY D3PLOT **1.1**
1.1 Element Types 1.1
1.2 Types of results processed by D3PLOT 1.3
1.3 D3PLOT Representation of Elements and Other Entities 1.4
1.4 LS-DYNA output files processed. 1.6
Permitted gaps in family member sequences 1.10
2 RUNNING D3PLOT **2.1**
2.1 Starting the code 2.1
2.2 Selecting a graphics device. 2.2
2.3 If D3PLOT will not open a window on your display 2.3
2.4 Client/server graphics using X-Windows and OpenGL 2.4
2.5 Memory Management 2.5
2.6 Multiple Windows and Models 2.6
2.7 Checkpoint Files 2.20
3 USING THE D3PLOT SCREEN MENU SYSTEM **3.1**
3.1 Basic screen menu layout 3.1
3.2 Mouse and keyboard usage for screen-menu interface 3.2
3.3 Dialogue input in the screen menu interface 3.5
3.4 Window management in the screen interface 3.5
3.5 "QUICK PICK" Options. 3.7
3.6 "Tabs" for multiple graphics windows. 3.9
3.7 Customising the User Interface 3.9
3.8 Shortcut Keys 3.13
4 BASIC DATA EXTRACTION AND PLOTTING **4.1**
4.1 Reading results from an analysis. 4.1
4.2 Displaying geometry and results. 4.14
4.3 Controlling contouring of data plots: CONTOUR 4.57
4.4 Animation How to display, control, store and retrieve animation sequences. 4.76
4.5 STATUS Listing programme status 4.94
5 VIEWING CONTROL **5.1**
5.1 Dynamic Viewing (Using the mouse to change views). 5.1
5.2 Viewing Control Buttons 5.3
5.3 Options under Viewing menu 5.6
5.4 Special 3D graphics driver options (OpenGL). 5.13
6 USING "TOOLS" OPTIONS **6.1**
6.0 Introduction to main menu commands 6.1
6.1 BLANK “Blanking” controls the visibility of nodes and elements. 6.5
6.2 VOLUME_CLIPPING 6.10
6.3 DEFORM Deforming geometry. 6.19
6.4 CUT_SECTIONS 6.37
6.5 ENTITY Switching the display of entity categories on/off. 6.56
6.7 WRITE Listing numerical data to screen and/or file. 6.62
6.8 XY_DATA Drawing numerical data as XY plots and/or writing it to file 6.79
6.9 UTILITIES Miscellaneous utility functions 6.101
6.10 GROUPS: 6.136
6.11 Coarsen 6.140
6.12 T/HIS the D3PLOT <=> T/HIS link 6.145
6.13 Trace Node 6.154
6.14 User Data 6.161
7 IMAGES **7.1**
7.0 Creating static images and movies 7.1
7.1 Static file formats supported 7.5
7.2 Animation file formats supported and their attributes 7.7
7.3 LASER PLOTTING 7.11
7.4 Reading static images and movies 7.20
7.5 Watermarks 7.24

7.6 Print ... option (Windows platforms only) 7.25
8 COMMAND AND SESSION FILES 8.1
8.0 Introduction to Command and Session Files. 8.1
8.1 CFILE Invoking the command-file launcher box. 8.2
8.2 Recording “session” files. 8.2
8.3 Playing back “command” files. 8.5
8.4 Using the “launcher” box during recording and playback 8.6
8.5 More information about command and session files 8.7
8.6 Associating command files with Function keys 8.8
8.7 Running command files from the command line. 8.8
9 DISPLAY_OPTIONS: 9.1
9.1 BACK_FACES switch: Display of “back” faces of solid and thick shell elements 9.1
9.2 INTERNAL_FACES switch: Display of inside faces of solid & thick shell elements 9.2
9.3 LOCAL_TRIADS switch: Display of element local axes 9.2
9.4 MODEL_BOX switch: Displaying the model external dimensions 9.3
9.5 UNDEFORMED... menu Displaying the undeformed geometry 9.4
9.6 SPRING_SYMBOLS... menu: Setting the drawing style for springs and dampers 9.6
9.7 BEAM_SYMBOLS... menu: Setting the drawing style for beams. 9.6
9.8 BELT_SYMBOLS... menu: Setting the sizes of seat-belt and related symbols. 9.7
9.9 SPH Symbols. Managing SPH element display. 9.8
9.10 AB Pcle Symbols: Managing Airbag Particle display 9.9
9.11 Other Symbols: controlling further symbols 9.10
9.12 HIDDEN_OPTIONS... menu: Setting hidden-line display options. 9.11
9.13 FREE_EDGES... menu: Controlling free edge display of element borders 9.15
9.14 WINDOW_DRESSING... menu: Controlling screen appearance. 9.20
9.15 Graphics text factor 9.22
10 PART TREE 10.1
10.1 Part Tree Behaviour 10.1
10.2 Part tree top menu bar 10.3
11 The Javascript Interface 11.1
11.0 Introduction 11.1
11.1 Using Javascript in D3PLOT. 11.1
11.2 The D3PLOT Javascript API 11.5
12 MORE ABOUT DATA AND DATA COMPONENTS 12.1
12.0 Introduction to this section on data and data components. 12.1
12.1 Format of the LS-DYNA databases processed by D3PLOT 12.2
12.2 Contents of the LS-DYNA database files processed by D3PLOT 12.6
12.3 Global (whole model) data components 12.13
12.4 Part ("material") data components 12.15
12.5 Contact Surface summary components 12.16
12.6 Nodal data components 12.17
12.7 Solid element data components. 12.21
12.8 Thin Shell element results. 12.25
12.9 Thick shell element results 12.32
12.10 Beam element results 12.36
12.11 Contact segment results 12.37
12.12 Smooth Particle Hydrodynamic (SPH) Data components 12.41
12.13 Airbag Particle (ABP) data components 12.42
12.14 Data components for other entity types 12.44
12.15 Theory and Formulae 12.45
13 D3PLOT USE OF GRAPHICS HARDWARE 13.1
13.1 The "X" (X_Windows) 2-D protocol. 13.1
13.2 3D protocol: OpenGL. 13.2
13.3 Summary of capabilities of each graphics protocol 13.3
14 PROBLEM SOLVING 14.1
14.1 Problems reading files: 14.1
14.2 General graphics problems: 14.3
14.3 Memory consumption problems. 14.4
14.4 Graphics problems peculiar to X_Windows devices. 14.4
14.5 Graphics problems peculiar to OpenGL 14.7
14.6 Miscellaneous problems. 14.8
14.7 MEMORY Viewing and controlling the memory usage for this process, and the whole machine. 14.8
Appendices A.1
APPENDIX I PROGRAMME LIMITATIONS A.1
APPENDIX II OA_PREF FILE: SETTING USER PREFERENCES A.2
APPENDIX III CHANGED DEFAULTS THAT AFFECT APPEARANCE A.11
APPENDIX IV COMMAND LINE OPTIONS AND WINDOWS FILE ASSOCIATIONS A.13
APPENDIX V ENVIRONMENT VARIABLES USED BY D3PLOT A.17
APPENDIX VII DIALOGUE COMMAND SYNTAX A.26
APPENDIX VI JAVASCRIPT API A.47

0 Preamble

Acknowledgements

The names **LS-DYNA**, **LS-PREPOST** and **LS-OPT** are all registered trademarks of the Livermore Software Technology Corporation (LSTC) and are used in this manual by permission.

Abstract

Transient analyses make much more sense when one is able to see how the results change with time. Most existing post-processors only allow you to draw one image at a time and, while it is possible to assemble a sequence, producing a set of results can be very tedious.

This code allows you to access the LS-DYNA database directly and to draw line, hidden-line, continuous-tone, line contour, velocity arrow, greyscale and shaded-image plots for any results state in the file. It also allows you to store these images in the display device memory and to redraw them in sequence and so to produce animated graphics.

Host computers

The code is available for all commonly used operating systems: Windows, Unix and Linux in 32 and 64 bit modes. It is available on all common work-stations and mainframes.

Memory requirements

Memory is allocated dynamically, so the amount required rises in proportion to the problem size. However machines with less than 64 MBytes of physical memory (RAM) are unlikely to function satisfactorily.

Output devices

The code supports the following graphics devices:

OpenGL	3-D, hardware assisted graphics
X_Windows	2-D unaccelerated graphics

Images may be captured in the following formats:

Animated "movie" formats:

AVI	AVI animation files
MPEG	MPEG animation files

Static "image" formats

BMP	BMP (bitmap) static image files
JPEG	JPEG static image files
PNG	PNG (Portable Network Graphics) static image files
GIF	GIF (Graphics Interchange Format) static image files
Postscript	Colour and greyscale laser plotting
PDF	Colour and greyscale Portable Document Format (PDF) output

External animations and static images may also be imported for display in the following formats:

BMP }
JPEG } For display of static images
PNG }
GIF

AVI For display of animated images

New Features for Version 9.3RC1

Static and Movie "image" output has been greatly enhanced:
- PNG and GIF formats have been added to the library of static image formats that may be written
- The MJPEG codec has been added to AVI movie output format, giving smaller and better quality files.
- Colour palette optimisation has been added for 8 bit image output, giving better colour distribution
- Images can be captured at 2x and 4x screen resolution if desired.

Laser file plotting has been improved and extended.
- Laser images are now captured directly from the display (like GIF, JPEG, etc), rather than by generating a 2D vector-style plot.
- As well as traditional Postscript (.ps) format it is now possible to write Portable Document Format (.pdf) format files.

New capability to read in static and movie images for display have been added:
- BMP, GIF, JPEG and PNG format static images may be displayed as static window background.
- AVI files may be displayed as animated window background, or in separate windows, synchronised with model animations.
- A "watermark" capability to display transparent images on top of the current image has been added.

A new capability to define, store, compute and display "user-defined" data components has been added.
- Any number of nodal or element user-defined data components may be created.
- These may be scalar, vector or tensor; and they may have arbitrary names.
- They may come from any permuation of the following sources:
  - Read from externally generated data file
  - Calculated internally from existing components via "simple formulae"
  - Calculated externally from existing components or other data via a Javascript interface
- The simple formulae and Javascript files can be saved and re-used across different analyses

The ability to select display via contour bands has been added:
- Clicking on a contour band will display just results for that band
- It is also possible, via a right-click popup menu, to display by other functions of contour band.

External data ("blob") plots can now be defined at nodes as well as at [x,y,z] coordinates.

Vectors of displacement and acceleration can now be displayed. (Previously this was only possible for velocities)
In addition if user-defined vector components are created at nodes these too may be displayed as vector arrows.

New Features for Version 9.2

The user interface has been completely redesigned to be more consistent with other Oasys products, giving a common "look and feel", in particular:
- Most common function panel function now occupy a tabbed working area on the right of the screen
- Keyboard short-cuts have been added for many functions
- The "entity" panel has been completely redesigned to conform to Oasys PRIMER layout
- Screen selection has been improved with many unnecessary button clicks eliminated.
- In general fewer button clicks should be needed to drive the software.

A "Part tree" has been added, giving a tree-style display of model contents, and most graphics functions may be invoked from the tree. If a .ztf file (from Oasys PRIMER) is present then this tree will also show information about include files and assemblies.

The cut-section function has been enhanced as follows:
- The main panel layout has been condensed and simplified, making it much quicker and easier to define cut sections.
- Dragging of sections has also been simplified, with rapid access to the most common operations.
- Optional dynamic feedback of current section forces when dragging has been added.
- A new "LS Dyna Method" way of defining sections has been added, with input identical to the *DATABASE_SECTION card
- If a .ztf file is present cut section definitions may be imported directly from the LS-DYNA model input deck.
- The force calculation method for "basic space" sections has been revised to use LS-DYNA's approach.

Automatic mesh coarsening has been added to speed up the display of large shell element models. If used the process is

fully automatic, and graphics speed increases of 2x to 3x or better can be achieved at the expense of a slight degradation of image quality.

Rendering has also been speeded up internally to give faster performance on some hardware where long runs of contiguous "mesh strips" can be processed quickly.

A new "external data" function has been added which will superimpose externally generated data on current images. This is normally used to add information such as externally calculated model parameters as discrete symbols on plots.

A new "trace nodes" function has been added to allow the historical path of nodes to be plotted through the states in an analysis.

A new "Iso surface" plotting mode has been added which plots contours of constant value in 2D and 3D meshes.

A new "Cloud" plotting mode has been added to represent data by point symbols on items.

The existing "SI" (Shaded Image) plotting mode now defaults to solid banded contours with lighting, but the original gouraud shaded ("fuzzy") contouring method is still available.

The XY plotting panel has been enhanced as follows:
- A new "Data vs Data" extraction and plotting mode is provided, plotting X vs Y components over time.
- A data sorting function has been added for the "Composite" plotting mode, allowing curve points to be re-ordered
- Control over filenames (.cur files) has been improved.
- The XY plotting tool itself has been improved, with better file saving syntax and a sorting tool for curves.
- Curve data transfer to linked T/HIS (when active) has been speeded up.

Data transfer via the ".ztf" file (ex Oasys PRIMER) has been enhanced to give:
- The ability to visualise node-based contacts (only segmented contact geometry appears in the .ctf file)
- The ability to extract and draw "true" beam sections

Limited support for ALE analyses has been added, with the ability to process components by name and also to visualise the surface geometry of ALE parts. This is work in progress ...

The Oasys T/HIS interface has been improved. It is now far more robust and better integrated with D3PLOT and, in particular, can handle multiple models in a more compatible way.

New Features for Version 9.0

The ability to display multiple models in windows is added, removing the "one model per window" limitation in V8.3. Significant changes to the user interface have been made to accomodate multiple model handling.

The general ability to process multiple models is greatly enhanced, and in particular models may be overlaid or artificially separated in a window, and they may be given different colours and display styles to distinguish them.

The ability to compare results across models (reference model/state) is added, allowing the differences between models to be computed and displayed. Models that are compared in this way do not have to be identical, but they do need to be reasonably similar if the results are to make sense.

Groups have been totally rewritten, removing the limit of 30 groups and provided automatic storage of those defined. A new ascii group file format, which is common to T/HIS and Primer, is also added. Backwards compatibility with old group files is retained.

Extra options have been added to the user interface making it more configurable, in particular font sizes, type face and left-handed support are now all configurable.

New Features for Version 8.3

Ability to display multiple windows added: up to 20 windows may be displayed concurrently.

Ability to handle multiple models added: up to 20 models may be processed concurrently.

Link with Oasys T/HIS added: automatically opens T/HIS, extracts time history data and permits full T/HIS processing. Time-history items can be visualised.

Native PC graphics: both 3D (OpenGL) and 2D graphics now use native Windows drivers, no longer any need for X11

emulators.

Improvements to cut-sections: better capping display of 2D elements, and sections may be dragged interactively with the mouse.

Programme start-up speed improved: time to first plot and first run through animation sequence approximately halved.

"Settings" file added: saves and retrieves layout and settings of multiple windows.

Function keys F1 .. F12 may be programmed with command files.

Optional automatic read of model properties file at model input time.

"Checkpoint" file added: saves all commands verbatim for replay, and aids crash recovery.

New Features for Version 8.2

Results database modified and improved:
- Reduced memory usage on Unix/Linux platforms via optional caching of data
- Data extraction and averaging speed improved.
- Animation speed improved.

Shell integration point data extraction and plotting now includes min/max/magnitude over all points through thickness

JPEG (still image) and MPEG (animations) file output added, and dithering added to 8 bit-plane BMP files to improve quality.

User manual (this manual) converted to HTML, with direct access via links in the HELP messages in the code itself.

"Feature" lines added as wireframe, hidden-line and overlay option.

More "oa_pref" and command-line options added - see appendices II and IV.

New Features for version 8.1

New static database added that contains the following data items. 1.4

Nodes on NODE_TO_SURFACE contacts

Spotweld Beams (section 6.4.7)

Nodal constaints and restraints (sections 6.5.1-6.5.2)

New Features for version 8.0a

Description	Section
"Quick-pick" blanking and attribute change has been added.	3.6
The blanking menu has been revised.	6.1
The properties file (prp file) in the props menus is now model independent.	3.1
Changes to the layout and appearance of the d3plot window have been made.	
The viewing "zoom" command now uses the "one-touch" method.	
Command line options and windows file associations App IV	

Manual Revision History

Date	Revision	Description
May 2001	0	Original release of Version 8.1 manual
Nov 2002	1	Manual updated for Version 8.3 Beta 3
Nov 2003	2	Manual updated for Version 9.0
May 2006	3	Manual revised and updated for Version 9.2
May 2007	4	Manual revised and Updated for Version 9.3RC1

Text conventions used in this manual

TYPEFACES:	Three different typefaces are used in this manual:
Manual text	This typeface is used for text in this manual.
Computer type	**This one is used to show what the computer types. It is also used for equations etc.**
Operator type	`This one is used to show what you must type.`
Button text	This one is used for screen menu buttons (eg **APPLY**)

NOTATION: -	Triangular, round and square brackets have been used as follows:
Triangular	To show generic items, and special keys. For example: **<list of integers> <filename> <data component>** **<return> <control Z> <escape>**
Round	To show optional items during input, for example: **<command> (<optional command>) (<optional number>)**
Square	And also to show defaults when the computer prompts you, eg: **Give new value (10) :** **Give data component (FX_AXIAL_FORCE) :** To show advisory information at computer prompts, eg **Give terminal type [M for list] :** **D3PLOT_MANAGER >>> [H for Help] :** Also to show implicit commands, eg **[WRITE] SCAN <entity> <number of values>**

1 LS-DYNA FEATURES SUPPORTED BY D3PLOT

1.1 Element Types

The following entity types in LS-DYNA can be processed in D3PLOT:

Solid Elements	8 noded "bricks" (hexahedra), and the elements they degenerate to: "wedges" and tetrahedra.
Thin Shells	4 noded quadrilaterals and 3 noded triangles.
Thick Shells	8 noded shells.
Beams	2 noded beams.
Discrete elements	Springs, dampers, lumped masses.
Seat-belt elements	Belts themselves, slip-rings and retractors.
SPH elements	Smooth Particle Hydrodynamic "sphere" elements (V9.3 onwards)
Airbag Particles	The particles used in the Airbag Particle inflator method (V9.3 onwards)
Contact surfaces	LS-DYNA models impact, friction, sliding, etc by the use of contact surfaces that can be thought of as two-dimensional elements overlaying the surfaces of solid and/or shell models. These are not true elements, but rather sub-areas, so to prevent confusion they will be referred to as interface "segments" from now on.
Using the .XTF file from LS-DYNA	In LS-DYNA joint, lumped-mass and stonewall geometries are sent to the **`.XTF`** file and so may be recovered for plotting in D3PLOT. However results from these are not available for plotting in D3PLOT: they may be viewed in XY plot form in Oasys T/HIS.
`Alternatives to the .XTF file when using MPP LS-DYNA`	MPP LS-DYNA, and also SMP versions from ls970 onwards can also generate a "binout" (or LSDA) file; and the MPP version cannot generate a **`.XTF`** file. D3PLOT does not read this file directly, but from V90 onwards the information previously extracted from the **`.XTF`** file is now available from the **`.ZTF file`** - see below.

Using the .ZTF file

Oasys PRIMER can generate a pseudo-database .ZTF file directly from the input deck. This is intended to contain extra information not in the normal LS-DYNA database files, and also to replace the .ZTF file.

In V8.3 the **ZTF** file allows you to visualise:
- Nodes on "nodes_to_..." contacts.
- Nodal constraints and restraints
- Spotweld beams.

From V9.0 onwards you may also visualise the following even when the .XTF file is missing:
- Stonewalls (rigid walls)
- Springs and dampers
- Seatbelt elements, retractors, slip-rings and pre-tensioners
- Joints
- Lumped masses

In addition the names of parts and contacts, previously stored in the .XTF file, are also available.

From V9.2 onwards you may also visualise:
- Beam "true" sections
- Part tree: organised by include files, assemblies and sub-assemblies (as in Oasys PRIMER)

From V9.3 onwards:
- Discrete and Seatbelt elements can be processed by PART
- Part, Part_composite and Section data are available

In addition ls-dyna cut-section definitions (*DATABASE_CROSS_SECTION) may be used to defined D3PLOT cut sections

If the T/HIS link is invoked then elements and nodes in time-history blocks can be displayed, and screen picked for time-history plotting.

Data for other entities are not sent to any database files, so they are not displayed.

D3PLOT is primarily for post-processing results from LS-DYNA, but results from TOPAZ3D (thermal analysis) and NIKE3D (implicit structural analysis) may also be processed. Both codes write a subset of the entity types listed above.

1.2 Types of results processed by D3PLOT

LS-DYNA is a three-dimensional non-linear analysis code which models the transient behaviour of structures in the time domain. The output you may process graphically in D3PLOT is shown by category below, together with the intrinsic coordinate system in which it is written by LS-DYNA.

ENTITY TYPE	DATA GENERATED		COORDINATE SYSTEM
Nodes:	Coordinates Velocities Accelerations Temperatures		Global cartesian " " " " [none]
Solids: (1)	Stress tensor Plastic strain Strain tensor (optional)		Global cartesian [none] Global cartesian
Thin shells: (2)	Stress tensor Strain tensor (optional) Forces and moments Plastic strain Thickness Strain energy density		Global cartesian " " Element local [none] [none] [none]
Thick shells: (2)	Stress tensor Plastic strain Strain tensor (optional)		Global cartesian [none] Global cartesian
Beams: (3)	Forces and moments Plastic data (optional) Stress/strain data (optional)		Element local " " " "
SPH elements: (4)	Stress tensor Plastic strain Strain tensor Density Pressure Internal energy Radius of influence	Also: #neighbours	Global cartesian [none] Global cartesian [none] " " " " " "
Airbag particles: (4)	Mass Radius Spin energy Trans energy Distance to nearest segment Coordinates Velocities	Also: Gas ID Leakage state	[none] " " " " " " " " Global cartesian Global cartesian
Contact surfaces	Contact stresses Contact forces		Segment local Global cartesian

(1) Results for solids are written by LS-DYNA at the element centre only, even if an element formulation with > 1 integration point is used. Solid results from NIKE3D are written at all 8 integration points.

(2) By default shell stress and strain tensor results are written at top and bottom integration points, and stresses also at the neutral axis. Data output at more than these 3 points through the element thickness may be selected, and will be available for display if present. Fully integrated shells in ls-dyna with more than 1 integration point on plan still only write (averaged) data at the element centre.

(3) As well as basic forces and moments extra "plastic" data from resultant beams, and data at integration points for integrated beams are supported.

(4) SPH and Airbag particle data are only processed from D3PLOT release 9.3 onwards.

D3PLOT will generate derived data components, (eg von Mises, principal, etc) from the above, and will also transform results from global to local coordinate systems if required.

In addition the following written output may be generated for more "global" model data.

ENTITY TYPE	DATA COMPONENT	COORDINATE SYSTEM
Whole model	Average velocity & momentum Kinetic and internal energies Mass	Global cartesian [none] [none]
Each material	Average velocity & momentum Kinetic and internal energies Mass	Global cartesian [none] [none]
Contact surfaces	Summary forces	Global cartesian
Airbags (of Airbag particles)	Volume	[none]
Stonewalls	Normal force	Local vector

Any data component written for nodes or elements that is actually present in the database files may be plotted graphically, presented as an X-Y plot of <data> vs <time> (or vs <data>), and written out in tabular form. This is also true of components derived from the basic ones.

In addition many geometric and topological attributes of nodes and elements (eg material number, elements connected to nodes) may be tabulated.

Any scalar data component may be "scanned" for maximum / minimum values, and tables of the top and bottom values produced.

Although their topology is extracted and they are displayed visually the results for springs, joints, seat-belts etc, are not available for plotting in D3PLOT. This is because these data are not currently available in the appropriate database files.

1.3 D3PLOT Representation of Elements and Other Entities

The three figures below show examples of how each of these types appear as drawn by Oasys D3PLOT. They also show the labelling conventions used:

Entity Type		Labelled
Solids Thin Shells Thick Shells		**H...** **S...** **T...**
Beams Springs / Dampers Seat-belts Slip-rings Retractors Lumped-masses		**B...** **SP..** **SB..** **SR..** **RT..** **LM..**
SPH Elements Airbag Particles	These elements may be drawn as points, cubes or spheres. This is controlled in Display Options	**HP..** **AP..**
Contact Segments Stonewalls Joints		**I...** **W...** **J...**
Nodes		**N...**

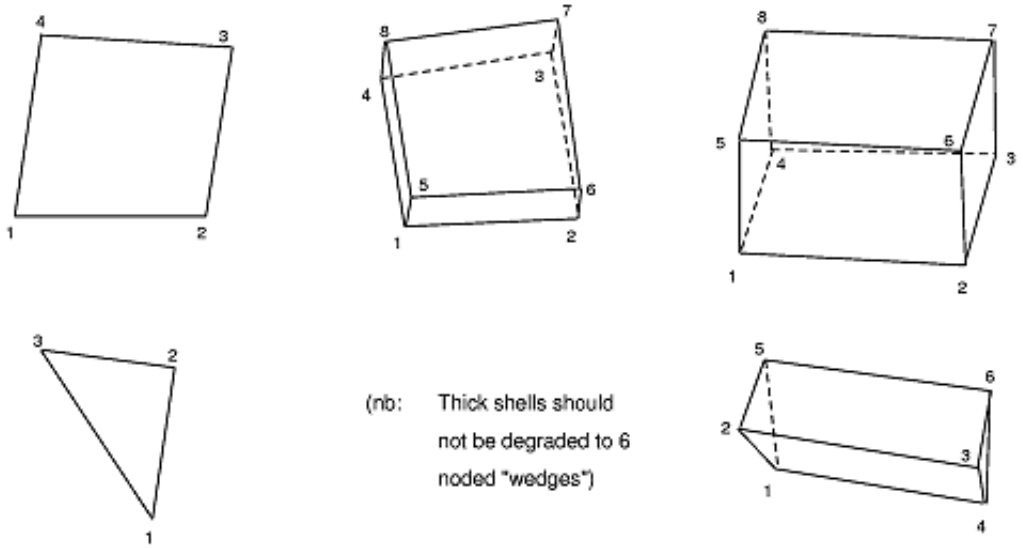
Note the following:

a) Arbitrary numbering of nodes, elements and materials in LS-DYNA is supported. This covers nodes, solids, shells, beams, springs, seat-belt types and lumped-masses. Joints, stonewalls and contact segments are all numbered sequentially from 1.

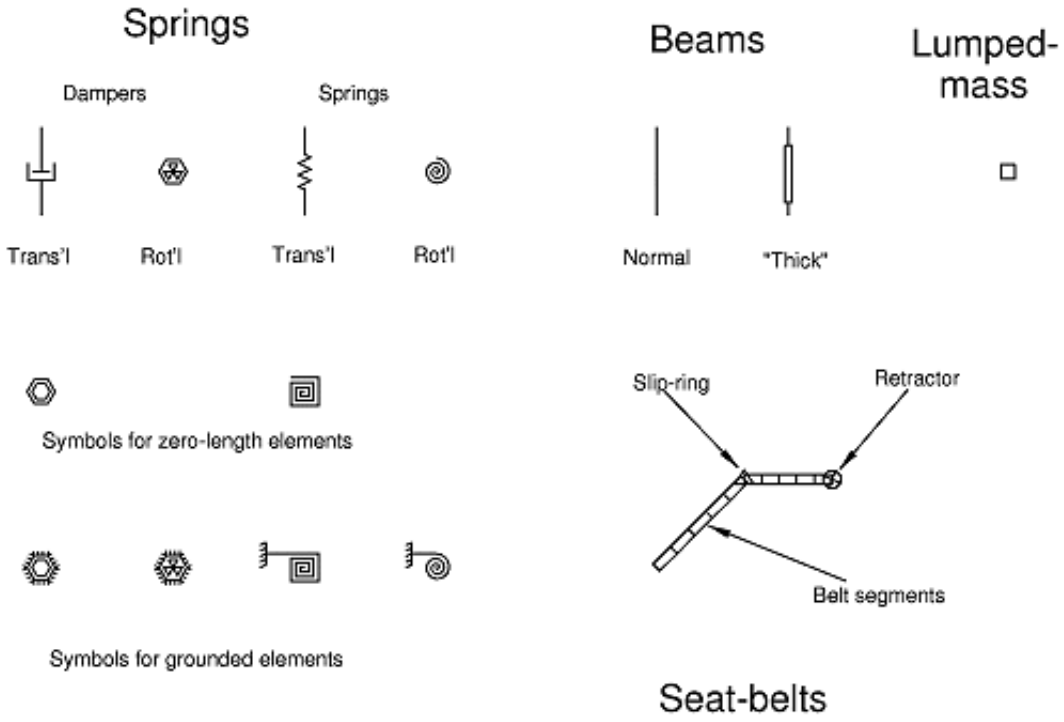
b) Springs, seat-belt types, lumped-masses, joints and stonewalls are only recovered and drawn if an "extra time-history" (**.XTF**) file is found - this file is optional. Only the topologies of these elements are extracted: use Oasys T/HIS to extract and plot time-history results for these elements.

c) Contact segments are only recovered and drawn if a "contact force" (**.CTF**) file is found - this file is optional.

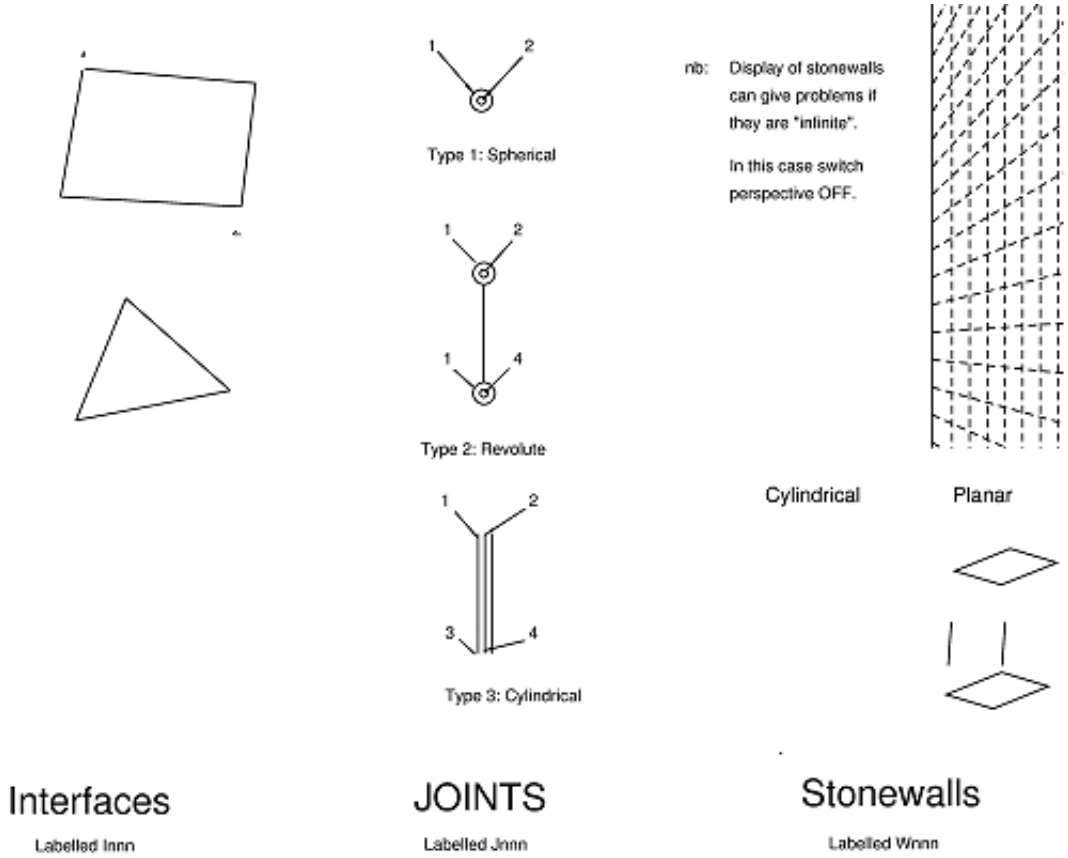
d) These figures show the symbols used on 2D devices. When 3D graphics is used some symbols are slightly different: springs become a spiral, damper symbols become a three-dimensional dashpot, joint circles become spherical, "thick" beams have rectangular sections. This is done to make symbols meaningful regardless of how the view is oriented in 3D space.



Solids shells and thick shells



Springs, beams, lumped masses and seatbelt elements



Contacts, stonewalls and joints

The next section describes in more detail which element categories are found in which files.

1.4 LS-DYNA output files processed.

D3PLOT reads the Oasys LS-DYNA binary database files directly. No intermediate translation is required and results can be viewed while an analysis is running.

The database files processed are (see Section 12.2 for more details):

Complete state (plot) file

`<name>.ptf`

This file contains the undeformed geometry of the model, followed by complete dumps of its current geometry. It contains information about:

Nodes:	Coordinates, velocities, accelerations, temperatures.
Solids:	Stresses, strains, (extra data).
Beams:	Forces, moments, plastic rotations, plastic strains.
Thin shells:	Stresses, strains, force & moment resultants, strain energy density, thickness, (extra data).
Thick shells:	Stresses, strains, (extra data).
"Global" data:	Material energies, masses and velocities; normal force on stonewalls.

The file also contains information about deleted elements if the relevant material models and/or contact surfaces are used.

Extra time-history file

<name>.xtf (See also under T/HIS link below)

This file contains miscellaneous "time-history" data about the model. D3PLOT reads only the following basic topology and coordinates from it:

Springs:	Spring, damper & seat-belt geometry.
Seat-belt types:	Seat-belt, retractor and slip-ring geometry.
Lumped-masses:	Geometry and mass.
Joints:	Geometry and type.
Stonewalls:	Geometry, mass and topology.

This file also contains the names of parts and contacts, which will be displayed in menus if available.

Contact force file

<name>.ctf

This file contains information about contact surfaces:

Contact facets:	Topology, contact stress.
Nodes on facets:	Contact forces.

Extra “static” database file

<name>.ztf

This file is not generated by LS-DYNA itself, but rather by running Oasys PRIMER on the relevant input deck. (This can be done automatically from the Oasys shell).

It is a "static" file (ie no time-history data) that contains information about:

Nodes on NODE_TO_... contacts	The nodes on the "nodal" side of these contacts will become visible as "diamond" symbols. However no force data is recovered on these nodes.
Nodal restraints & constraints	For each of the six degrees of freedom of every node any restraint or constraint due to SPCs, *CONSTRAINED items, rigid bodies, inclusion on TIED or CONSTRAINT contacts, etc is stored. These DoFs can be displayed.
Beam section data	By storing beam element section data it becomes possible for D3PLOT to "know" which beams are spotwelds (ie section type 9 using *MAT_SPOTWELD) and to draw them as spotwelds.
Part and Section data	From V9.3 onwards the *PART(_xxx) and *SECTION_xxx cards are written verbatim to the .ZTF file, making it possible to extract thickness and layer information. In addition it becomes possible to associate the Part ids of Springs and Seatbelt elements with those used by solids, shells and beams making it possible to operate on these "by part".

From V9.0 onwards, where the .XTF may not be present, this file also contains all the information previously stored in the .XTF file so that there is no loss of functionality.

Files read using the D3PLOT <=> T/HIS link

<name>.thf
<name>.xtf
<ascii files>
<binout(LSDA) files>
If the optional D3PLOT <=> T/HIS link (see section 6.12) is used then D3PLOT gains access to the time-history data embedded in these files. (Ordinarily access to the .XTF file above only extracts geometry data).

The link permits nodes and elements in time-history blocks to be visualised and selected graphically for time-history processing.

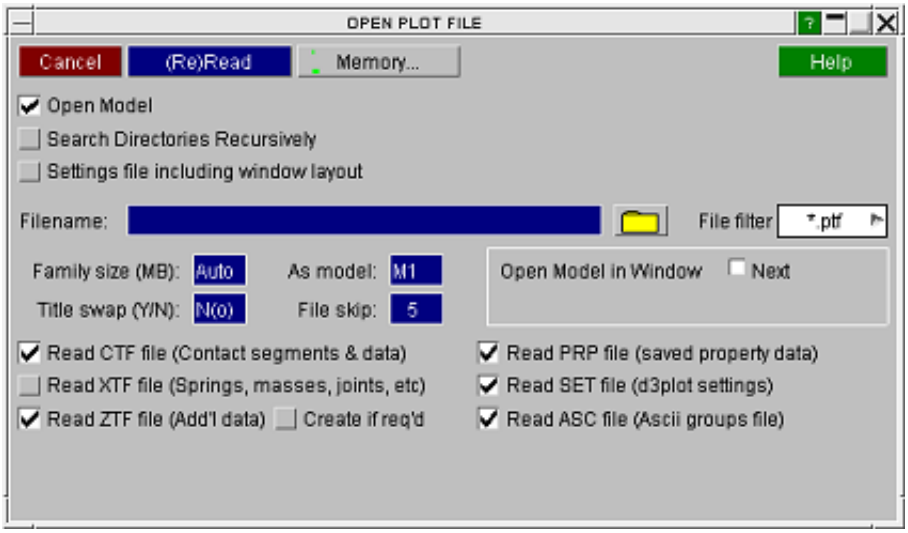
Files required for D3PLOT to run.

D3PLOT must have a complete state (**.ptf**) file in order to function. If any of the other files (**.xtf**,**.ctf** and **.ztf**) are missing then the entities within them will not be processed.

You can choose whether or not read the **xtf**, **ctf** and **ztf** files by checking the relevant boxes on the front file selection panel.

Here the contact force file ("**Read CTF file**", etc) has been de-selected, but the other two database files will be read if present.

In V9.3 a **ztf** file can be created automatically if required if the input (.key) deck is available by ticking **Create if req'd**.



Three further, optional files generated from previous D3PLOT runs may be read in: (none of these files is required)

The "**PRP**" file contains model properties, written from the **PROPERTIES** panel.

> This is a model-independent file of element and node properties that can be applied to the current model. "Properties" are colour, transparency, blanking status, etc. See section 6.9 for more information about this file.

The "**SET**" file containing saved D3PLOT settings written from the **UTILITIES, SETTINGS** panel.

> This is programme-specific data, allowing virtually all the options on the user interface to be saved and restored. For example the number and layout of windows, current data components, etc. See section 6.9 for more information about this file.

The "**ASC**" ascii groups file written from the **GROUPS** panel.

> An ascii groups file is a compact and human readable file of group information that can be applied to any model.

Converting between binary file formats.

In version 8.0 onwards of D3PLOT conversion of binary file formats to the native type of the processing computer is fully automatic: database files may be generated on one computer and post-processed on another with an incompatible binary format and/or precision. Specifically the conversions between:

- 64 to 32 bit word lengths.
- Cray to IEEE numeric representation.
- Big to little endian organisation

are performed automatically without any input from the user.

Database filename syntax

D3PLOT supports all the following database filename syntax options from current and previous releases of Oasys LS-DYNA:

	"Old" syntax (pre release 6.0)	**"New" syntax, release 6.0 onwards**	**Default filenames if none defined**
Database filenames	**<name>.ptf**, **.p01** **<name>.xtf**, **.x01** **<name>.ctf**, **.c01**	**<name>.ptf, .ptf01** **<name>.xtf, .xtf01** **<name>.ctf, .ctf01** **<name.ztf>, (n/a)**	**d3plot, d3plot01** **xtfile, xtfile01** (none) **ztfile**, **n/a**
Permitted #chars	4 in <name> 3 in <ext>	<name> + <ext> any number less than 80	n/a
Max #family members	99	999	999
Pathname permitted	No	Yes	Yes

Binary file family member size

In D3PLOT 8.0 onwards the determination of binary file family member size is automatic by default. D3PLOT takes the actual size of files, rounded up to the nearest Mbyte, as being the effective size for a given family and no further intervention by the user is required.

This can be overridden by giving an explicit size when the file is opened, or subsequently, or by setting the environment variable **FAM_SIZE** to an explicit size in MB. For example on a UNIX system:

setenv FAM_SIZE 1 (1MByte family, C shell syntax)

FAM_SIZE=9, export FAM_SIZE (9MByte family, Bourne shell syntax)

Any family size is legal, but it is suggested that it be a multiple of 1MByte on single precision machines. On machines generating double precision (64 bit) output files the same numbers (ie 1 and 9 in the examples above) should be used, but the actual file sizes will be 2 and 18 MBytes respectively. This is covered in more detail in Section 12.1.2

Permitted gaps in family member sequences

It is possible to skip over gaps in file family member sequences. The "file skip" variable may be set when files are read in, and when the code is running. This is covered in more detail in Section 12.1.3.

It is also possible to change this globally on UNIX systems by setting the **FILE_SKIP** environment variable. For example:

setenv FILE_SKIP 5	(Skip 5 files, C shell syntax)
FILE_SKIP=9, export FILE_SKIP	(Skip 9 files, Bourne shell syntax)

On some installations these variables are set globally for all users in the **Oasys_81** shell - consult your system manager.

Hint: On UNIX systems you can list all environment variables in the current shell with:

printenv

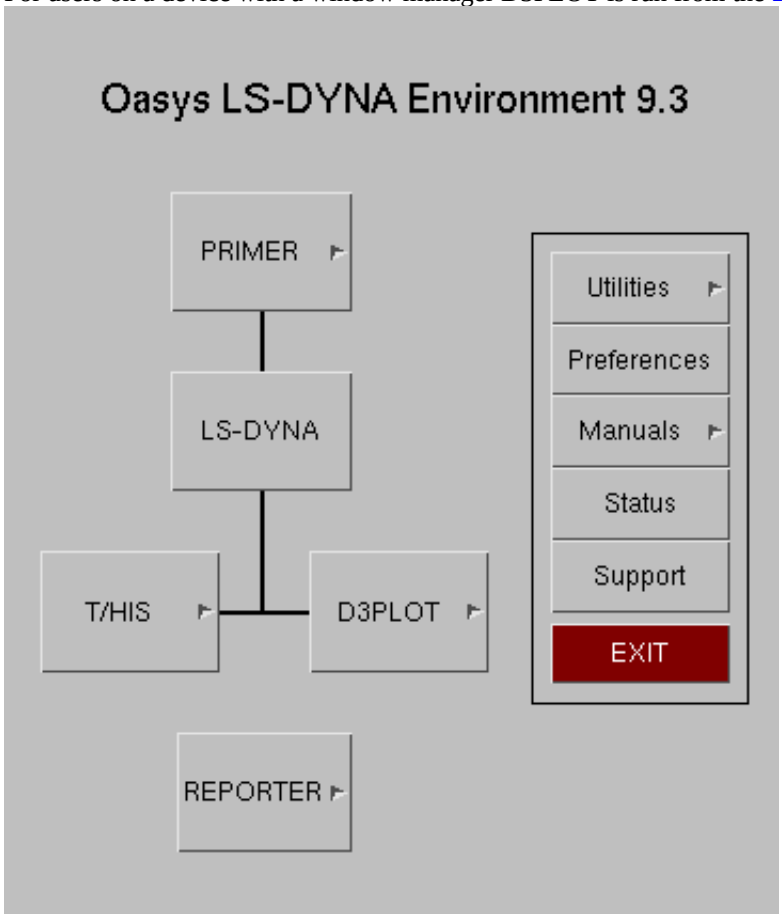
On Windows systems use **Control Panel**, **System** to view and set environment variables.

If they seem to be set correctly, but don’t seem to be affecting your process, remember that such variables must be set before the process starts. This is because a child process inherits properties of its parent when it starts, but thereafter is autonomous. You may need to exit and restart the process to make them take effect.

2 RUNNING D3PLOT

2.1 Starting the code

For users on a device with a window manager D3PLOT is run from the **D3PLOT** button in the **0ASYS_93** shell:



Users who are running on a device without a window manager should use the **PL** option in the command-line **0ASYS_93** shell.

Users on Windows platforms may associate the filetype ".ptf" with D3PLOT if they wish, so that double-clicking on a .ptf file starts the code. The way to do this is defined in Appendix IV.

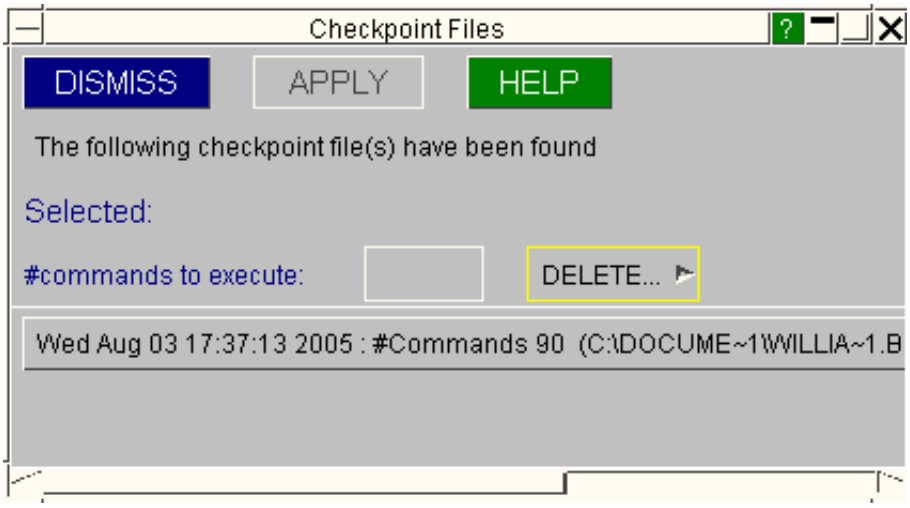
If your system has been customised locally you may have to use some other command or icon: consult your system manager in this case.

2.1.1 The Checkpoint File panel.

A "Checkpoint" file saves all commands and mouse actions during a D3PLOT session. It is deleted when the code terminates normally, but is left on disk if a crash, system failure or some other abnormal termination occurs.

If you see the checkpoint file panel when the code starts this means that a previous D3PLOT run has crashed, and recovery is possible.

Checkpoint files are described in section 2.7

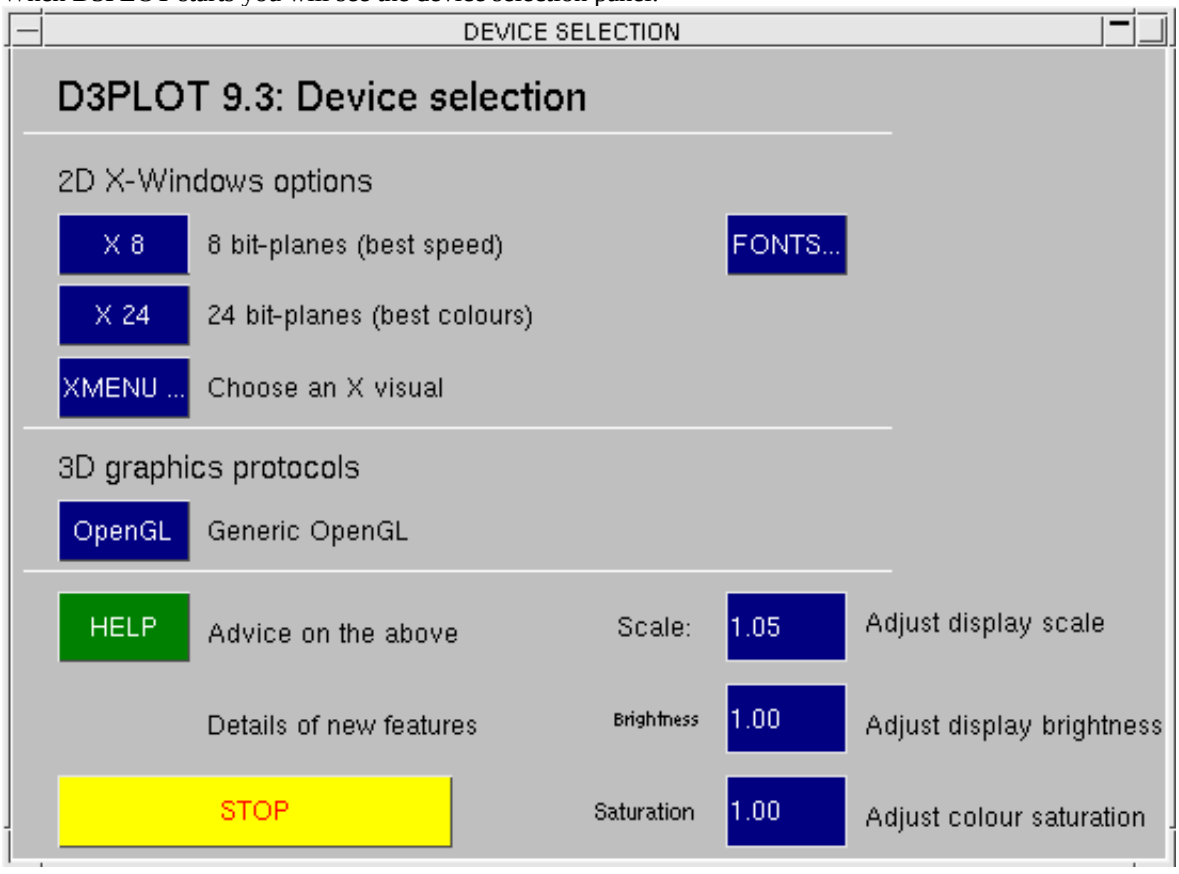


2.2 Selecting a graphics device.

You will not see this panel on a Windows platform, where OpenGL is automatically selected.

On a Unix/Linux platform if your installation has been customised with a graphics device specified on the command-line ("**-d=<*device*>**") you will also not see this panel.
To customise startup see Appendix IV.

When D3PLOT starts you will see the device selection panel:



The actual devices available will depend on your machine type and the graphics options that have been installed. The graphics drivers available are:

X8	2D X-Windows graphics at 8 bit-planes depth. This will be faster to animate than X24 below, but shaded image quality will be mediocre (using only 256 colours).
X24	2D X-Windows graphics at 24 bit-plane depth. Slower to animate than X8, but shaded image quality will be good (using 16M colours).
XMENU...	2D X-Windows under a user-defined visual. (See below)
OpenGL	3D OpenGL rendering, using hardware acceleration if available.

Scale **Brightness** **Saturation** **Fonts**	Controls the effective scale of the display used for the menu interface. Controls colour brightness in the menu interface Controls colour saturation in the menu interface Controls the fonts used in the menu interface.	In D3PLOT release 9.2 these display attributes can all be modified interactively using the **Options > Menu Attributes** popup panel, and this is the preferred method as the effects of changes can be viewed instantly and settings can be stored automatically in the "oa_pref" file. This is described in section 3.7

Which graphics option should I use?
The simple answers are:

You should use OpenGL if:
- Your machine supports it with hardware acceleration. Graphics will be faster than X-windows, and special features such as shading, lighting and transparency are also much better

You should use an X-Windows option if:

- OpenGL is not available (button greyed out).
- Your model requires too much memory to be processed under OpenGL (2D graphics are much less memory-hungry).
- You are working in client/server mode over a slow network. (The network traffic generated by 2D graphics is less)

From release 9 onwards the X driver is no longer being developed, and it will increasingly be the case that new features are not supported in it.

An alternative, using the MESA OpenGL emulator under X11 is available: please contact Oasys if you would like to use this option.

There are more complex answers to this question which depend upon:
- Model size vs. memory available.
- The rendering method chosen.
- The number of frames you plan to animate.
- Whether image quality or speed is more important.
- Local/remote machine types if in client/server modeNetwork performance in client/server mode

There is a longer discussion in Section 13, refer to this if you have problems.

2.3 If D3PLOT will not open a window on your display

If you get a message stating:

```
Could not open display <hostname>:0
```

and no window appears, you have failed to make a connection with the X11 server.

(Note that OpenGL also uses the X11 server, so this section is equally applicable to both X-windows and OpenGL graphics.)

This is almost certainly because of one or both of the following setup errors:

(1) The **DISPLAY** environment variable has not been set up, or has been set incorrectly, on the "client" machine (where the D3PLOT process is running).

This environment variable tells the X11 window manager on the client machine where to place windows, and it must be set to point to the screen you plan to use. Its generic Unix setup string is:

setenv DISPLAY <hostname>:<display number> (C shell syntax)

Where <hostname> is your machine's name or internet address, for example:

setenv DISPLAY :0 (Default display :0 on this machine)

setenv DISPLAY tigger:0 (Default display :0 on machine "tigger")

setenv DISPLAY 69.177.15.2:0 (Default display :0, address 69.177.15.2)

You may have to use the raw network address if the machine name has not been added to your **/etc/hosts** file, or possibly the "yellow pages" server hosts file.

(2) The machine on which you are attempting to open the window, the X11 "server", has not been told to accept window manager requests from remote clients.

This is often the case when you are trying to display from a remote machine over a network, and you get a message on the lines of:

Xlib: connection to "<hostname>" refused by server

Xlib: Client is not authorised to connect to server

In this case go to any window on the server with a Unix prompt and type:

xhost +

Which tells its window manager to accept requests from any remote client. It will produce a confirmatory message, which will be something like:

access control disabled, clients can connect from any host

Networked graphics are a complex topic: see Section 13 for more detailed advice if the remedies here don't work. Alternatively see your system manager, or contact Oasys for advice and help.

2.4 Client/server graphics using X-Windows and OpenGL

It is relatively common to display 2D X-windows graphics from remote hosts on a local server. However it is less well known that exactly the same mechanism can also be used to display 3D graphics under OpenGL.

This can be a very efficient solution to the problem of rapid display of large datasets since the tasks of computing and displaying the graphics and, just as importantly, the memory consumption, are split over two machines. Both machines must be OpenGL compatible for this to work.

To do this:

Set the **DISPLAY** environment variable on the client to point to the server (2.1 above).

Start D3PLOT in the normal way and read in the model.

Select **OBJECT** display mode (States Box, **ANIM >, DISPLAY MODE, OBJECT**)

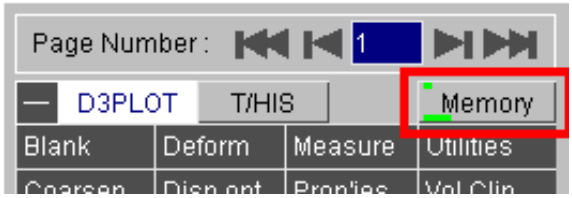
This will have the effect of storing all graphics as OpenGL "objects" in the server, making animation and redraw speed extremely fast. However keep an eye on memory consumption of the server: objects may be fast but they use a horrendous amount of memory.

2.5 Memory Management

Do I need to worry about this? Unless your machine is showing signs of running out of memory the answer is no.

There is a simple way to tell if this is the case:

If the **Memory** button bars are both green you have no problems. If either turn dark orange you may need to take some action.



The top one shows this D3PLOT process size as a proportion of available physical memory on the machine. If this exceeds about 85% (and goes dark orange) the performance of the code may start to degrade as it starts to page (use virtual memory, or "swap"), although it will continue to run.

The lower one shows swap space usage (by all applications) as a proportion of total swap space available on the machine. If this exceeds about 90% you may have to take action to free some space from elsewhere. (A machine with no free swap space will simply stop, and may need rebooting!)

There are alarms built into the code which will warn you if you are approaching either of these limits, so you don't have to keep checking memory consumption.

If a memory alarm pops up ...

- The memory use of this process can be managed by controlling how much data is stored in the results database, and also by changing the animation display method.
- Swap space is used by all processes on your computer, and it may be possible to free space by shutting down other processes.

A more detailed description of memory management, and the functions available under the description of the **MEM**(ory) button.

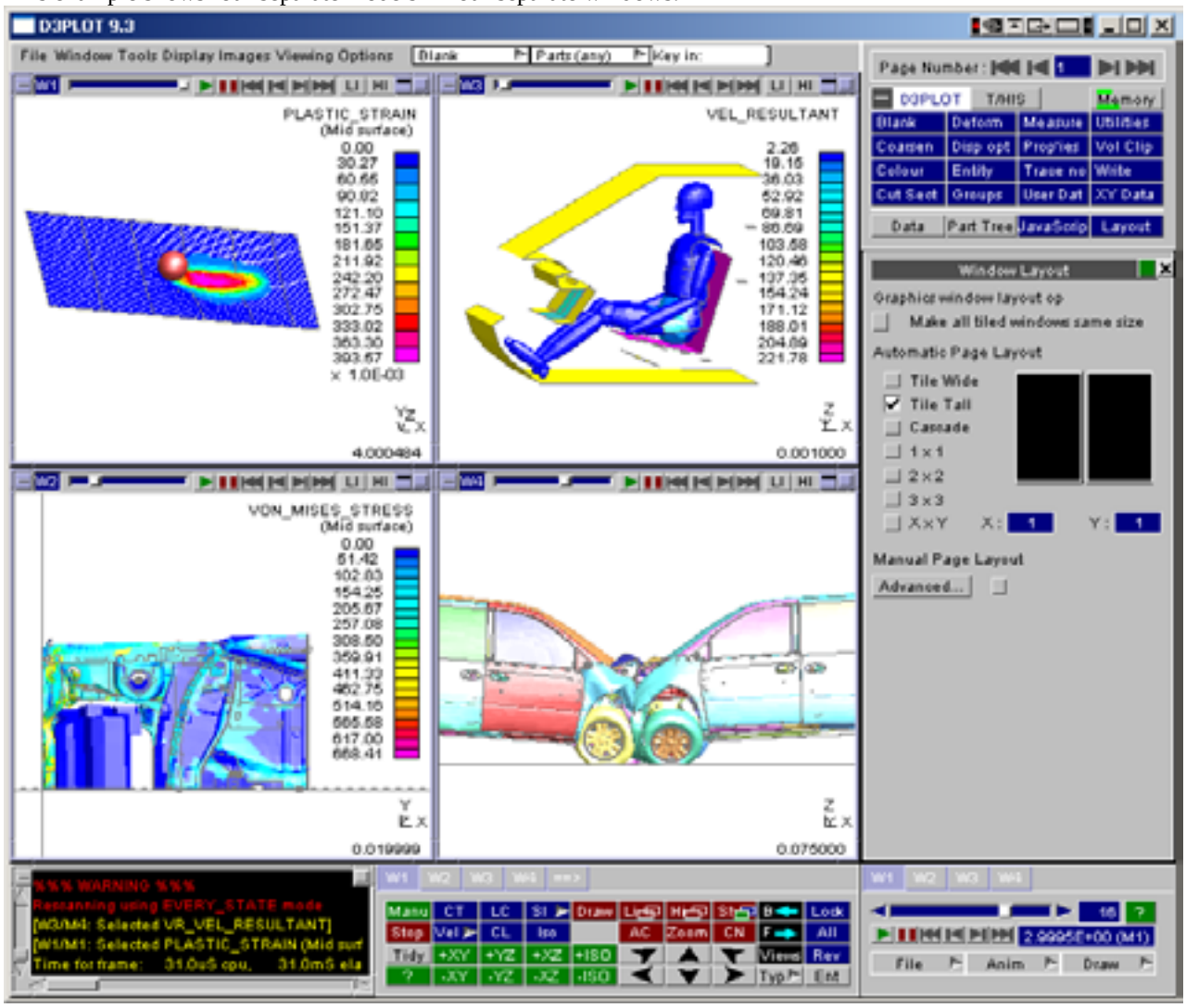
Click here for the next section

2.6 Multiple Windows and Models

From release 9.3 (Nov 2008) onwards D3PLOT supports the following permutation of multiple windows and models.

- Up to 32 windows may be defined.
- Up to 32 models may be current in memory.
- Any permutation of model(s) may appear in any window(s), subject to the limit of 100 "instances" of window/model combinations.
- Windows may be arranged on up to 32 "pages".

This example shows four separate models in four separate windows.

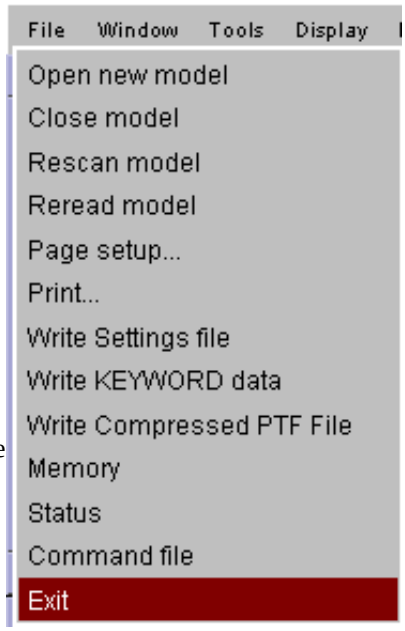


This example also shows how each window can have totally separate attributes: display mode, state number, view, background colour, etc. These can be controlled separately or collectively by using the "tabs" on each menu panel.

Where a window contains multiple models all that models in that window are given the same attributes (component, surface, etc); however it is possible to distinguish between models by:

- Separating them artificially in space
- Giving them different colours
- Drawing them in different modes (wireframe, shaded, etc)

2.6.1 **FILE >** Popup: Opening, Closing and Rereading Models



The **OPEN NEW MODEL** command maps the standard input file selection panel (see section 4.1)

Each new model will be opened in a new window. Up to 20 models may be held concurrently in the database, but you should note that they all compete for the same memory resources in the computer and that performance may be badly affected if you try to read in too much data.

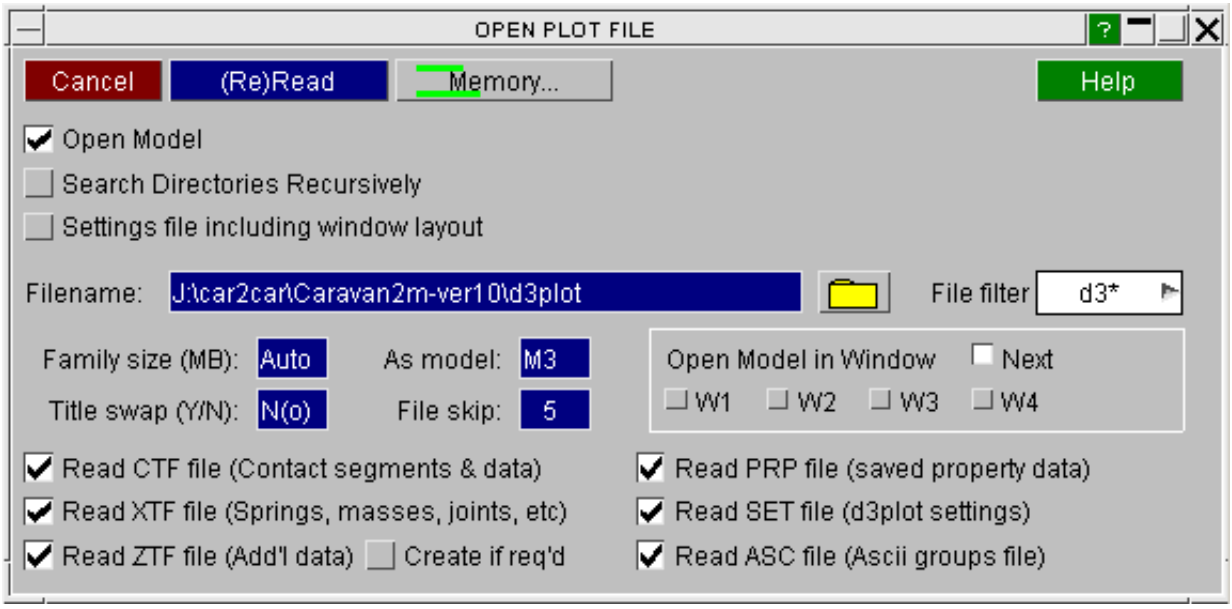
When you **CLOSE** a model all windows that display it only will also be closed, and the model's storage deleted from memory. You are warned before this happens.

RESCAN should be used when an analysis is still running and you want to search for more states.

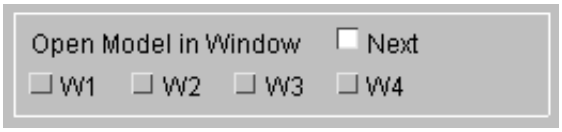
REREAD is equivalent to a Close/Reopen sequence: it completely rereads a model from scratch, and should be used if a model has been rerun. It should also be used when an adaptively remeshed file family has been extended

2.6.1.1 Choosing which window(s) to read a new model into.

The first model opened is always read into window 1, but models after that may choose which windows they will become active in.



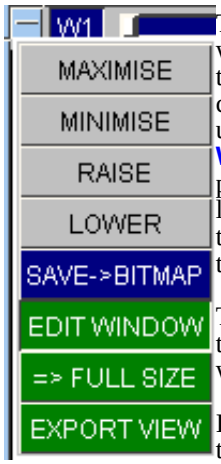
By default they are read into a new window, the "next" one, but you can select any other window(s) as destinations using the "In Window" buttons.



Any permutation of buttons may be selected, and the new model will become active in those windows.

You can subsequently activate and de-activate models in any window at will by using the **EDIT WINDOW** popup menu described below. This also allows you to separate models, set their drawing mode and also their colour.

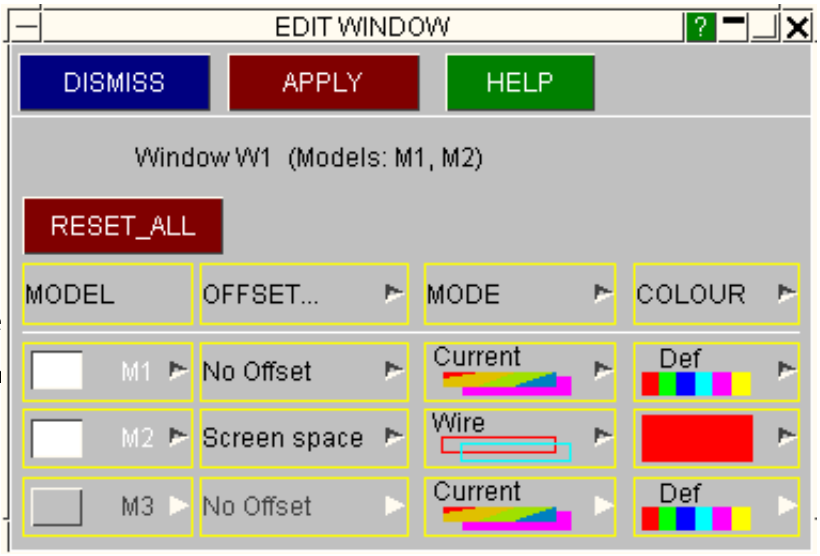
2.6.1.2 EDIT WINDOW Changing "Model in Window" attributes



The models active in a window, and some of their attributes, can be changed at any time using the **EDIT WINDOW** option in the popup linked to the top left **[-]** button, or from the window menu on the top bar.

This maps the panel to the right, here for window W1.

In this example 2 out of the 3 models are active in this window. Model M2 has been offset in screen space and is drawn wireframe in red.

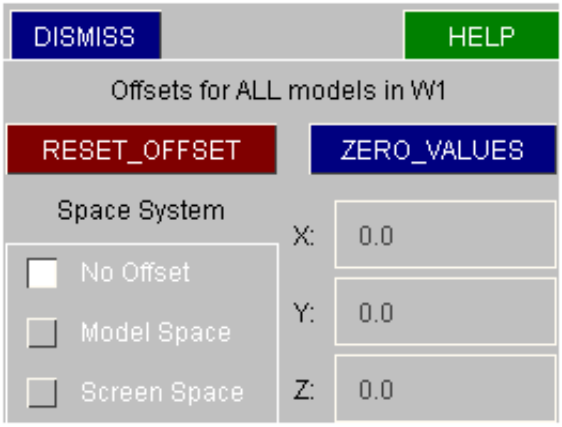


MODEL turning models on and off.

Simply toggle each model on and off. To see more attributes of the model right click to get its title and filename.

Note that if a model is not active in any window then it will be deleted from the database, you are warned and made to confirm this before it happens.

OFFSET... adding an artifical offset to models in windows.

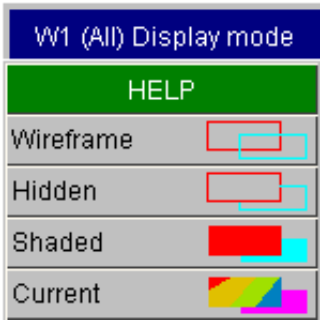


Multiple models in a window often overlay one another, and it can be useful to separate them. Right-click on the relevant OFFSET... button to map this panel, then choose offsets in one of:

- **`Model Space`**. Shifts the model in its own space system.
- **`Screen Space`**. Shifts the model in the plane of the screen.

You can visualise the difference between these two by considering how two dancers on a stage would react to rotation by 180 degrees when separated. In model space they would effectively swap sides of the stage; whereas in screen space they would each pirouette about their own toes, staying in the same positions.

MODE changing the display mode of a model in a window



Normally models are displayed in the current mode of the window, whatever that may be, but you can restrict them to:

WIREFRAME	No shading, hidden-surface removal or contouring; edges only.
HIDDEN	No shading or contouring, and edges only, but with hidden surface removal enabled
SHADED	Shading, but no contouring. Both edges and and lit surfaces displayed
CURRENT	Whatever the current display mode is.

The actual display mode used for a model will be **`min(current mode, mode selected here)`**. In other words selecting SHADED here will only produce a WIREFRAME plot if the current mode is only WIREFRAME (eg **LI**)

COLOUR setting a constant colour for a model in a window.

Normally a window in a model will be drawn using its normal colours, which are properties of the model itself.

You can override this by setting a constant colour for the model which will be used instead, which can be useful for distinguishing between two similar models that have been overlaid.

To return to normal colouring for the model use the

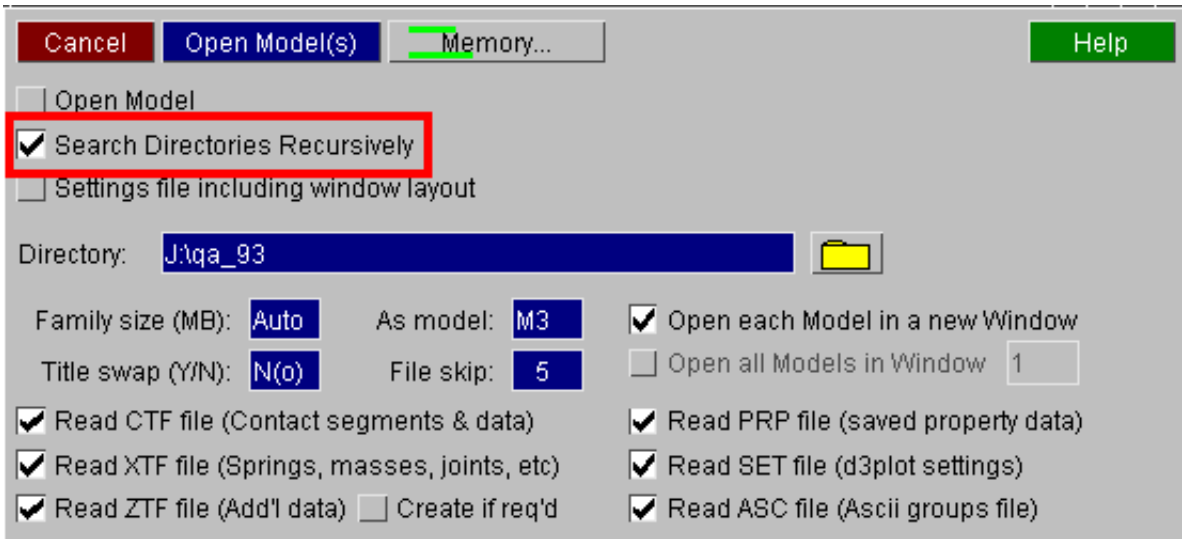
button.

(Note that you can achieve the same effect by changing the colour of elements in the model using the **PROPs** or **COLOUR** panels, but this will apply to all windows in which the model appears, whereas this option only affects the display of the model in this window.)

2.6.1.3 Opening Multiple Models in a Directory Tree

From V9.3 onwards it is possible to open up to 32 models simultaneously from a directory tree.

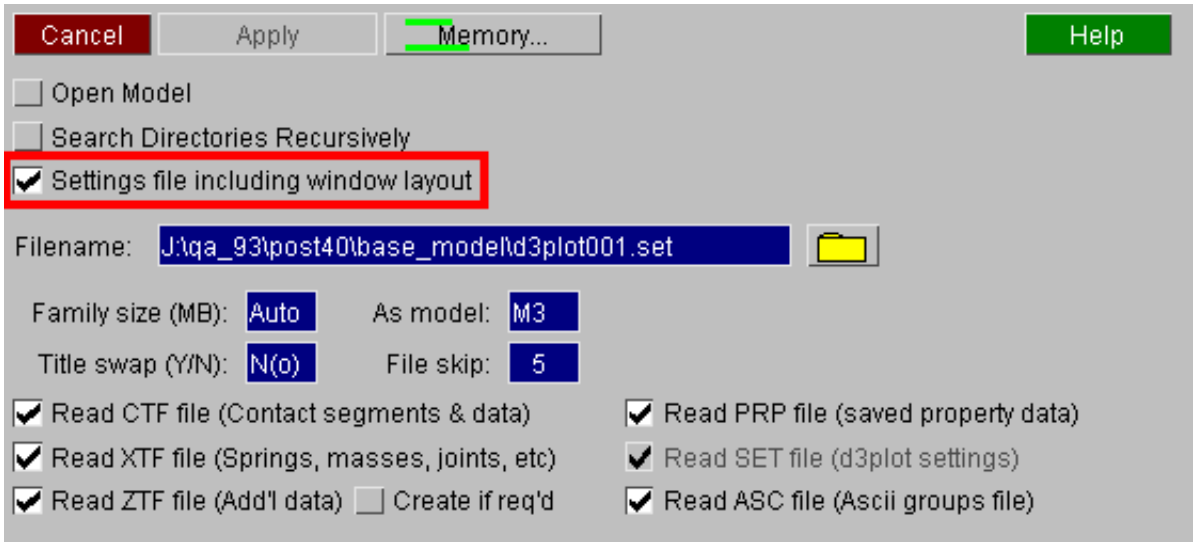
See **Search Directories Recursively** for more information on this.



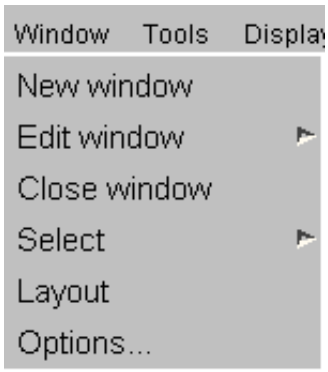
2.6.1.4 Opening Multiple Models using a Settings File

From V9.3 onwards it is also possible to open up to 32 individually chosen models with the attributes and layout specified in a Settings file.

See **Settings file including window layout** for more information on this.



2.6.2 **WINDOW >** Popup: Window management.



NEW WINDOW creates new windows. If there is more than one model in memory you have to choose the model to be placed in the new window. The newly created window will always be numbered as the next free one in the labelling sequence, and positioned in its default "layout".

EDIT WINDOW raises the window content editing panel described above once you have selected which window you want to operate on.

CLOSE WINDOW allows you to close any permutation of windows. If you close all the windows used by a model then that model is also deleted from memory. When a window in the middle of a sequence is closed the remaining windows above it are renumbered downwards so that there are no gaps in the sequence.

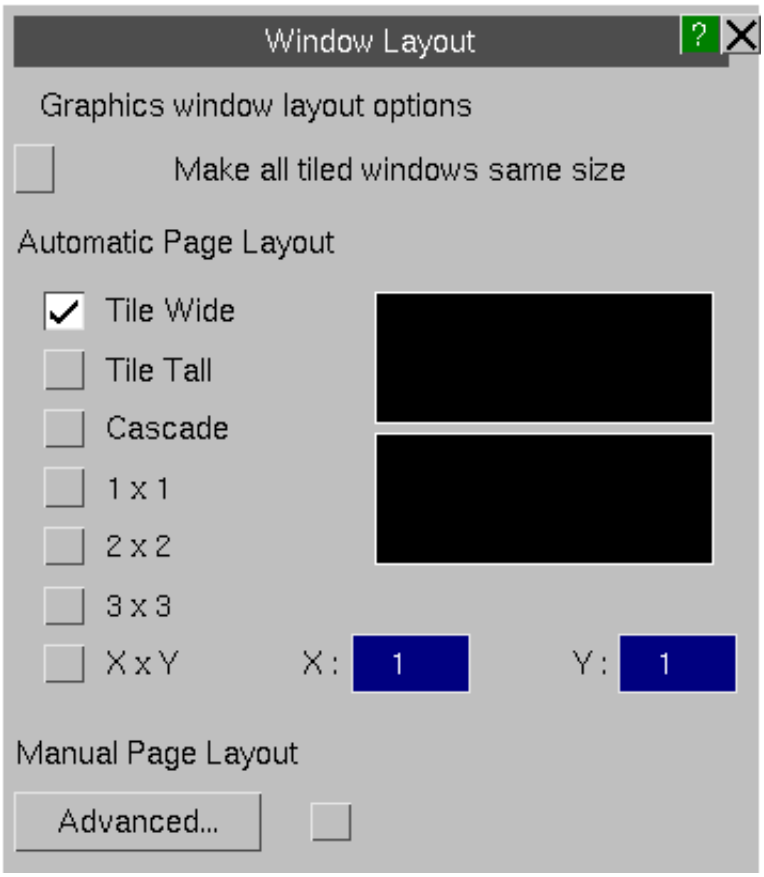
SELECT > is the "global" window tab selector. This topic is covered in more detail below.

LAYOUT... controls how multiple windows are organised on the screen.

OPTIONS... controls further aspects of window management and display.

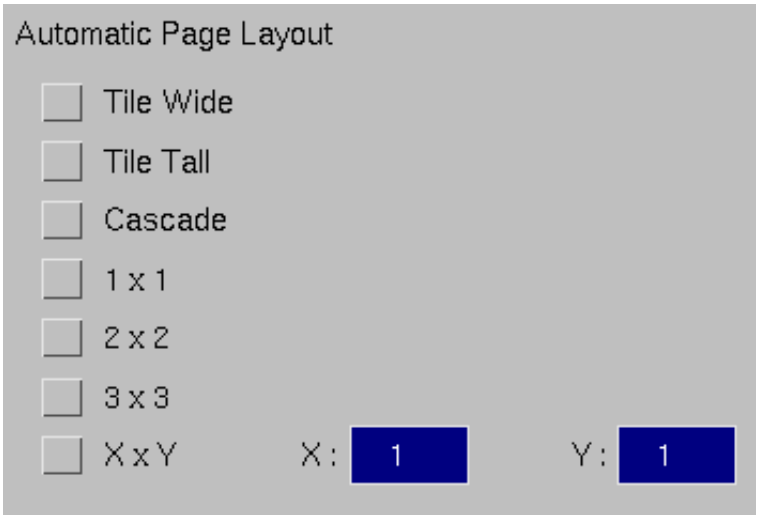
2.6.2.1 Window **LAYOUT...**

Windows can be laid out using a number of different formats and can be organised into 'Pages'.



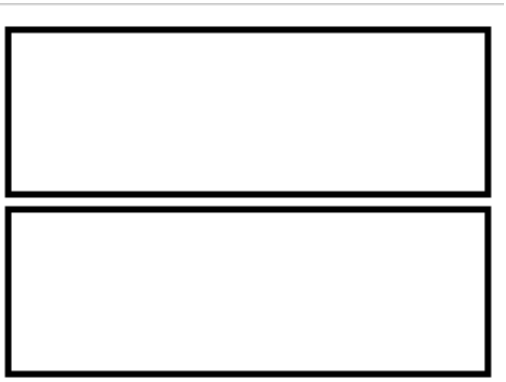
Automatic Page Layout

If an Automatic page layout is used and the layout is set to **1 x 1**, **2 x 2**, **3 x 3** or **X x Y** D3PLOT will automatically create multiple pages and position the windows on each page if required.

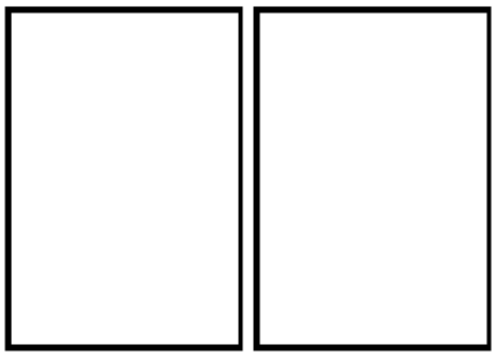


Tile Wide

All of the graphs are positioned on a single page.

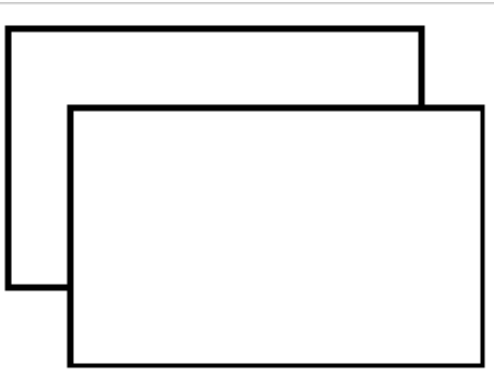


Tile Tall
All of the graphs are positioned on a single page.



Cascade

All of the graphs are positioned on a single page

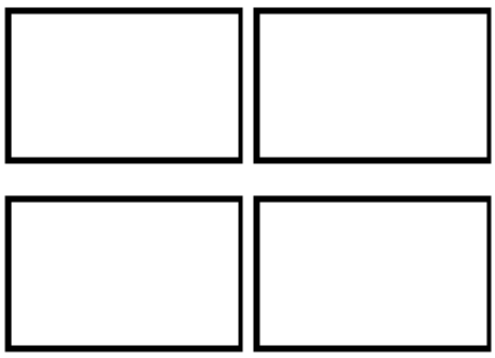


1 x 1

Each graph is positioned on it’s own page.

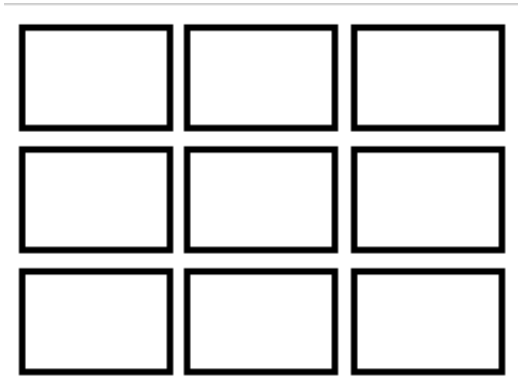
2 x 2

Graphs are arranged in a 2 by 2 grid. If there are more than 4 graphs then graphs 1 to 4 are positioned on page 1, 5 to 8 on page 2 ...



3 x 3

Graphs are arranged in a 3 by 3 grid. If there are more than 9 graphs then graphs 1 to 9 are positioned on page 1, 10 to 18 on page 2 ...



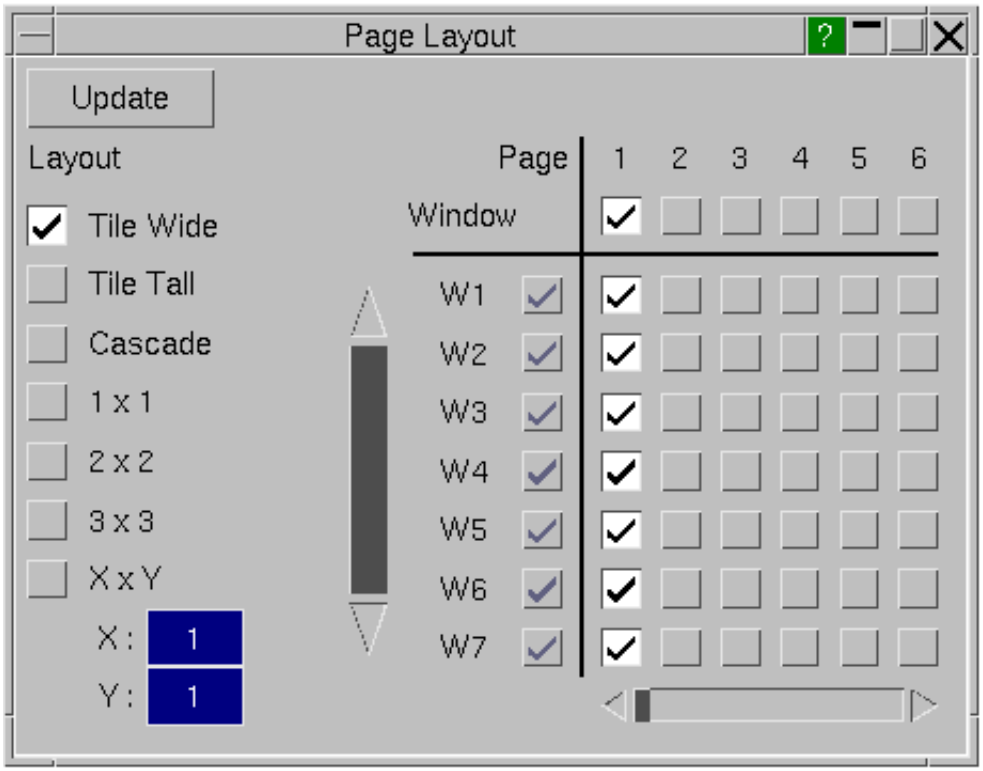
X x Y

Graphs are arranged in a X by Y grid.

Manual Page Layout

Manual page layout can be used to give more control over which graphs appear on which page. Unlike the Automatic page layouts a graph can appear on more than one page.

Advanced



The Advanced option displays the Page Layout menu. This menu can be used to select which windows appear on each page.

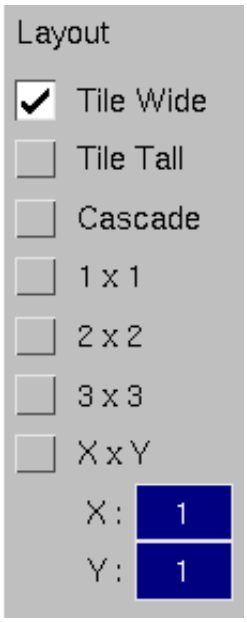
A range of windows can be added/removed from pages by selecting the first window/page combination and then holding down **SHIFT** while selecting the second window/page.

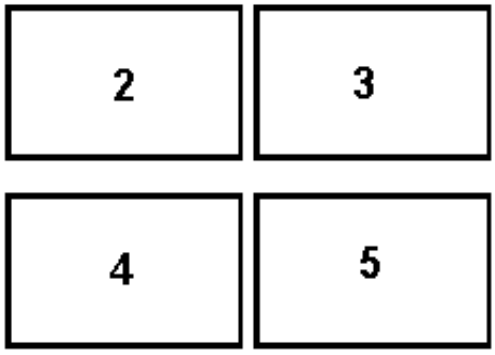
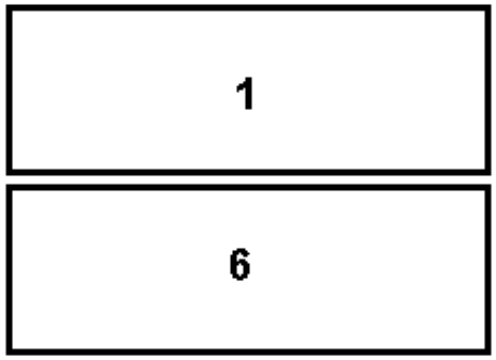
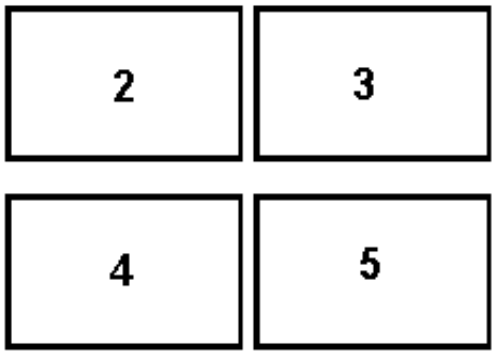
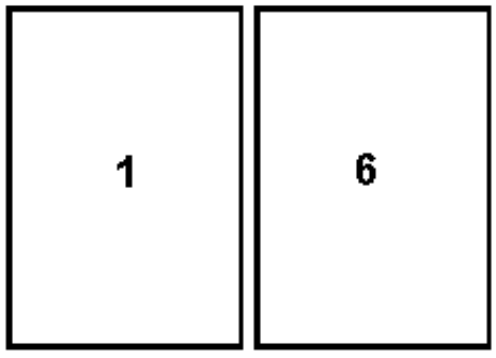
These buttons can be used to add a window to all pages.

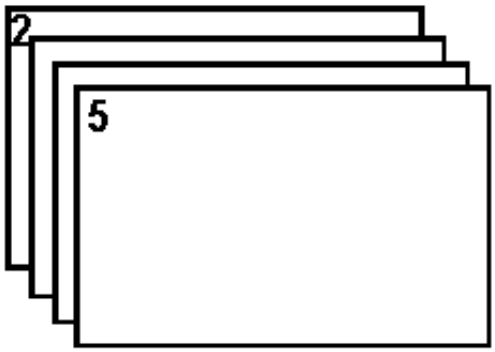
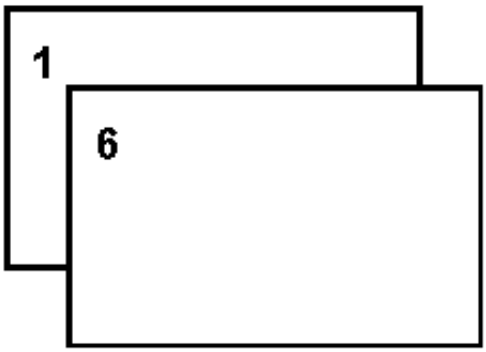
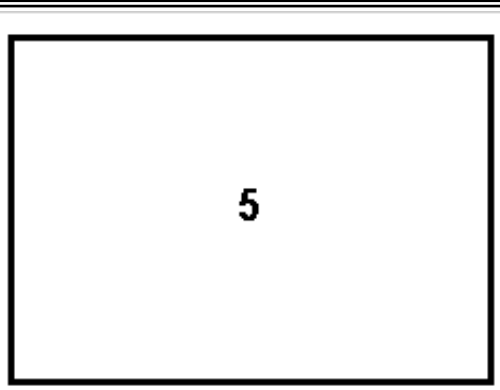
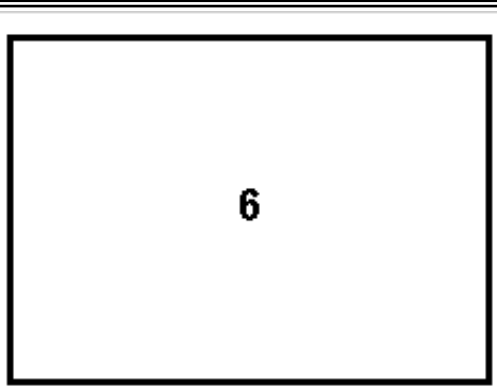
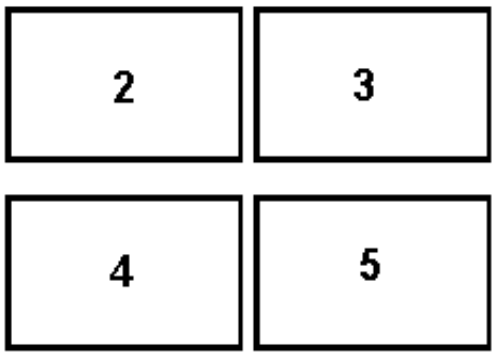
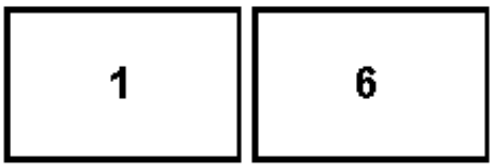
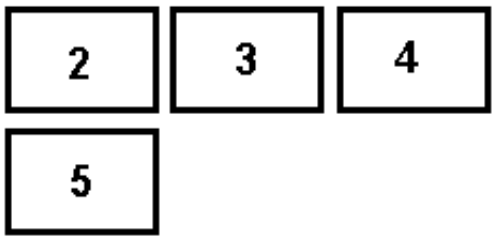
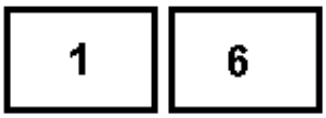
These buttons can be used to add all windows to a page.

The **Layout** options work in exactly the same way as the Automatic Page Layout options, except they only position the graphs defined on each page.

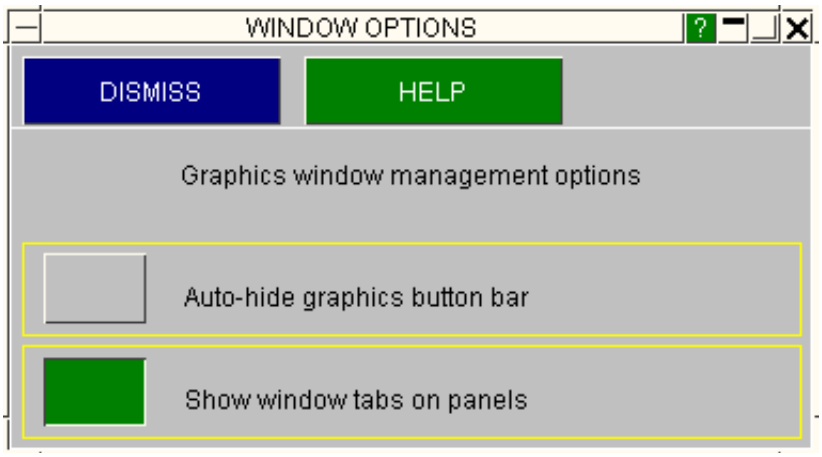
If for example D3PLOT has 6 windows defined and windows 2,3,4,5 are defined on page 1 and windows 1 and 6 are on page 2 then the different window layout options would produce the following.



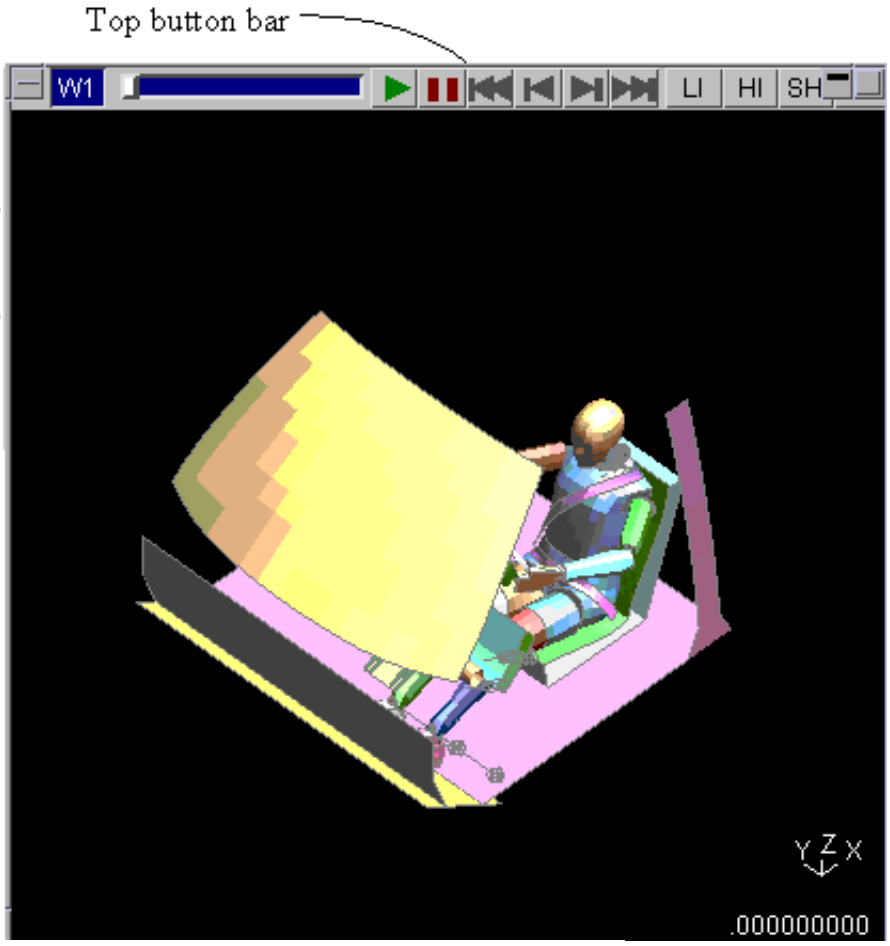
	Page 1	Page 2
Tile Wide		
Tile Tall		

Cascade		
1 x 1 (stacked on top of each other)		
2 x 2		
3 x 3		
X x Y	Layout depends on X and Y	Layout depends on X and Y

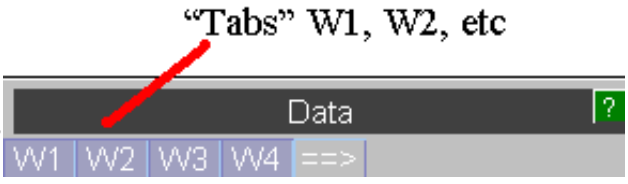
2.6.2.2 Window **Options...**



"**Auto-hide graphics button bar**" automatically maps the button bar at the top of each window when the cursor enters that window, and hides it again when the cursor leaves. This can be useful when you have many windows, or a small display, as it maximises the amount of space available for graphics. By default auto-hide is off, and the button bar is permanently displayed in all graphics windows.



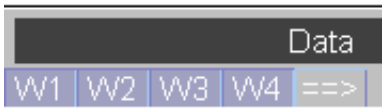
"**Show window tabs on panels**" controls whether or not the W1, W2, ... "tabs" for multiple graphics windows are displayed at the top of menu panels. If these are suppressed you will not be able to control the application of commands to windows on a per-panel basis. Window "tabs" are discussed in more detail below.



2.6.3 Controlling how commands apply to Windows.

When there is only one graphics window current then there is no ambiguity about where commands issued in menu panels apply, but once two or more windows are current the situation becomes more complicated. For example you may want to contour X stress in window 1, but Y stress in window 2.

To handle this problem D3PLOT automatically adds "tabs" (**W1**, **W2**, ...) for each graphics window to most menu panels when two or more graphics windows are present.



Using ***Wn*** tabs on menu panels.

Commands in menu panels will only apply to those graphics windows for which the ***Wn*** tab buttons are pressed. In the example above all four windows will be affected when commands are given in this panel. If a tab buttons is deselected then subsequent commands in this panel will not apply to that graphics window.

Some further rules apply:

- Each top level menu panel tabs selection is independent. Deselecting a tab button in one panel will not affect any other top level panels on the display.
- Selections propagate downwards to newly mapped children. For example if the **Component** panel is invoked from the **Current Operations** one it will initially inherit its parent's tab selection. However it is not limited to this and can subsequently be changed.
- The current status of a menu panel is influenced by its tabs setting as follows:
  - Where only one status word is displayable (eg component, shell surface) the first active window's value is shown.
  - Where status button is mixed (eg ON in window 1, OFF in window 2) then "ON" will be shown, but on a grey rather than coloured background.
  - If no tabs are active then the whole panel will be de-activated.

Propagating settings for a panel: the **==>** button.

Sometimes you may want to propagate settings from one window to others. This can be done in a limited way on a "per menu panel" basis using the **==>** button, which:

- Takes the settings for the first active window in that panel;
- Copies them to all subsequent active windows.

This is mostly commonly used in the View Control panel to make all windows have the same view as W1.

Using the ***Wn*** buttons on graphics windows.

Each graphics window has its number given in the ***Wn*** button at its top left corner. which can be used to toggle on/off that window's tabs in all current menu panels.

Here the **W1** button for window #1 is shown.

When the ***Wn*** button is toggled OFF:

- The border round the graphics window changes from light blue to grey
- The ***Wn*** tab in all active menu panels is toggled off, and the panels updated.

When it is toggled on again the opposite occurs.

This is simply a quick way of changing the tab status of all active menu panels: the window is not de-activated in any way, it can still be drawn in, and the local tab for this window can still be changed on any menu panel.

WINDOW > Select ... (De-)Selecting all windows.

It is possible to select and deselect all windows.

This is equivalent of toggling the ***Wn*** buttons on all graphics windows on (**ALL**) or off (**NONE**), and all menu panels will be affected.

2.6.4 What settings are "per model", not "per window"?

It will be obvious that different windows may present different views of a model, or contour different data components; but windows on the same model are not totally independent of one another. Storing detailed attributes for every item in a model on a per window basis would be wasteful of memory, and would also require some very complicated panels to provide detailed feedback to the user.

Therefore "properties" of a model, which can be stored in a properties file, are stored on a per-model rather than a per-window basis. This means:

- Blanking status; as controlled in the **BLANK** panel.
- Colour, transparency, brightness and shininess; as controlled in the **PROPs** and/or **COLOUR** panels.

"Properties" files are described in more detail under Properties.

If it is necessary to have multiple windows with different blanking or element attribute properties then you will have to read the same model in twice. D3PLOT will treat this as two totally separate models, and you will be able to set different attributes, however it will double your memory usage.

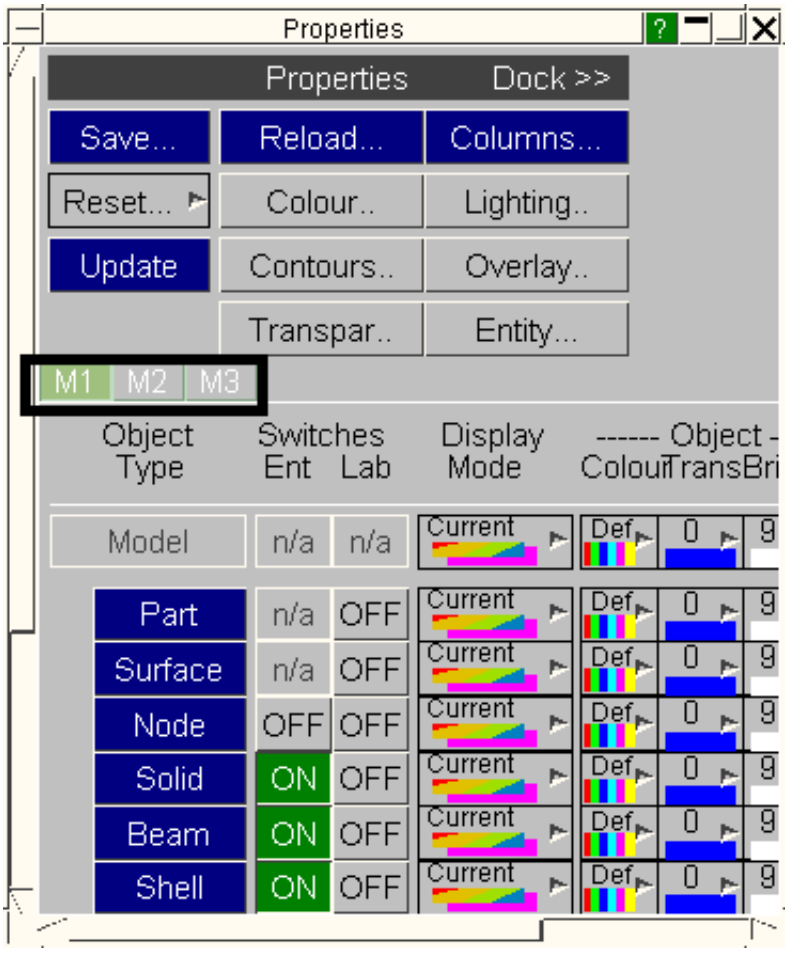
2.6.4.1 Swapping between models in model-specific panels

Model-specific panels can only operate on one model at a time, but you can swap between models at any time using **M1, M2, ...** tabs.

These function just like window tabs (**W1,** ...) except that:

- They are green rather than blue
- Only one tab may be active at a time

If you only have one model in memory then these tabs are not drawn.



2.6.5 Handling multiple models and multiple windows

Where multiple windows exist on a single model there is no possibility for conflicts between windows when commands are applied. However when multiple models have been read in lots of potential problems arise, to name just a few:

- Models may have different numbers of states, at different times.
- Some models may contain data components that don't appear in others
- Models may contain different types of element

D3PLOT has to protect itself against internal conflicts arising from attempts to impose invalid settings on windows, and also has to allow the user to manipulate multiple model settings in a simple way. So the following rules apply where multiple, dissimilar models are current:

- Where a panel would apply an invalid setting to a model no action is taken. For example turning on spring labels in a model that contains no springs will have no effect. Note that the feedback in a menu panel is generally for the first active window, so to see the status of the 2nd or subsequent models it may be necessary to adjust the tabs so that the model in question is the "first active" one for that panel.

- The **State Number** slider will show the highest state number of all the current models. If it is set to a state that doesn't exist in a given model then no action will occur in the windows of that model. Selecting state #n with this slider will make state #n in each window current, regardless of whether or not the times of this state in different models match.

- Contexts implying selection, eg **BLANK**, **WRITE**, **XY_DATA**, will force you to specify which model you plan to select from. The tabs in that panel which apply to other models will be deselected, and any attempt to select them will fail with "mixed model" error messages. Therefore you cannot pick or select from multiple models at the same time.

2.6.5.1 How models with dissimilar states are animated.

By default models are animated in numerical state sequence, with no attempt being made to synchronise models or windows by time. A summary of behaviour is:

- **Multiple models in separate windows**
  Each window animates in step from frames 1 to <n>, but no window will "loop back" to frame 1 until the window with the greatest number of frames has finished. Therefore those windows with fewer frames will wait at the end of each cycle for the window with longest sequence to complete. This "stepping together" by frame does ***not*** take into account the clock time of each frame, so windows that are "in step" by frame number will not necessarily be synchronised in time.

- **Multiple models in the same window:**
  Each model in the window starts animating at state #1, and continues until the last state in that model is encountered. If one model has fewer states than another one then it remains at its last state until the other model reaches its last state. Then all models start in synchronisation at state #1 again. Again, synchronisation is "by frame" not "by time", so frames in different models may not have the same analysis time.

- **Synchronising models in time.**
  It is possible to interpolate between states, and by stipulating a fixed time interval you can synchronise animation of multiple models in time. This can be done both for models within a window, and for models across multiple windows.

This topic is covered in more detail under the **SET_STATES** command, which describes how to select what is to be animated.

2.6.6 Comparing results between models.

It is possible to derive results in one model with respect to another - essentially by subtracting data in Model B from that in Model A. This is done in the **DEFORM, REFERENCE STATE/MODEL** panel. Briefly:

- You can plot data and coordinates relative to a state in the current model.
- You can also plot relative to a state (or the current state) in a different "reference" model.

For example you can plot the difference between two analyses where you have changed a section property, or remeshed an area.

Data is compared using labels, for example the data for node 100 in Model B is subtracted from that for node 100 in Model A. Therefore the models need ***not*** be identical, but they do need to be topologically similar for this to work. In particular comparisons in regions which have been remeshed are likely to be unsatisfactory.

In future releases we hope to perform "geographically" based comparison, where the node or element in Model B

nearest to that in Model A is used, removing the dependency upon identical labels.

2.6.7 Some special multiple window cases.

The **IMAGES** and **MOVIES** panel

Images may be made from one or more windows, according to which tabs are selected, see Images for more information.

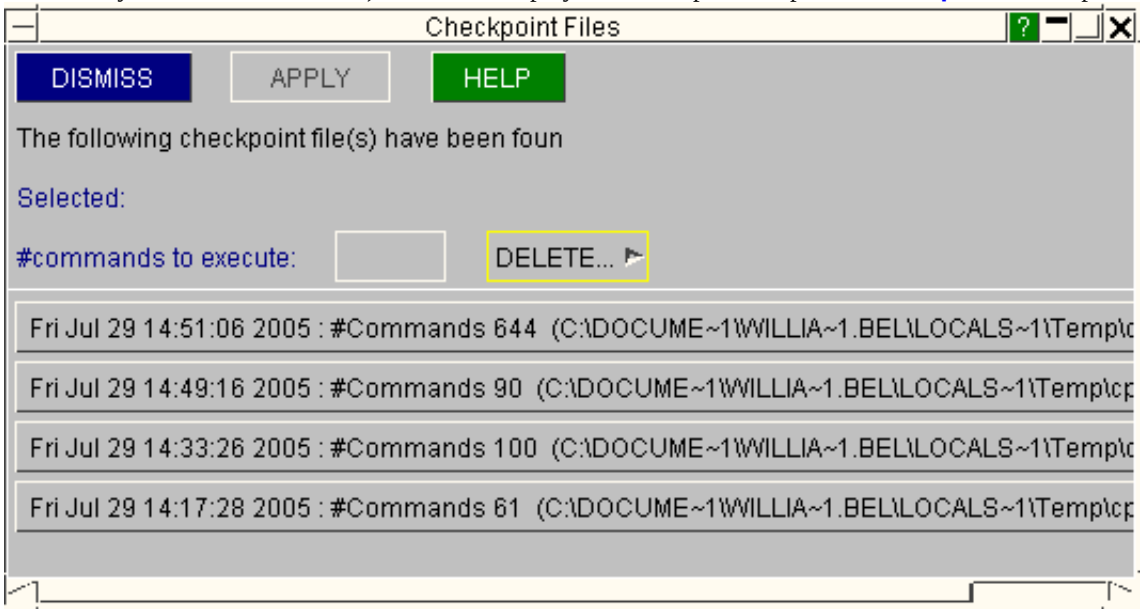
2.7 Checkpoint Files

From V8.3 onwards D3PLOT automatically records every command and mouse action in a "checkpoint" file. If the session terminates normally this is deleted, but if a crash occurs this file will be left on disk giving you the opportunity to recover your work.

Checkpoint files have the name "**`cp_D3PLOT_9.3_`***<pid>*" where *<pid>* is the current process id, ensuring that the filename is unique. They are normally written in the current directory, but if this is read-only they are written in $HOME or, failing that, in $TMP.

2.7.1 Selecting a checkpoint file.

When D3PLOT is restarted after a crash it automatically detects any checkpoint files in the current directory (or, if this is read-only, in $HOME or $TMP). These are displayed at startup in the special **Checkpoint Files** panel:


Checkpoint Files
DISMISS
APPLY
HELP
The following checkpoint file(s) have been foun
Selected:
#commands to execute:
DELETE...
Fri Jul 29 14:51:06 2005 : #Commands 644 (C:\DOCUME~1\WILLIA~1.BEL\LOCALS~1\Temp\c
Fri Jul 29 14:49:16 2005 : #Commands 90 (C:\DOCUME~1\WILLIA~1.BEL\LOCALS~1\Temp\cp
Fri Jul 29 14:33:26 2005 : #Commands 100 (C:\DOCUME~1\WILLIA~1.BEL\LOCALS~1\Temp\c
Fri Jul 29 14:17:28 2005 : #Commands 61 (C:\DOCUME~1\WILLIA~1.BEL\LOCALS~1\Temp\cp

All checkpoint files found are listed in date/time order, with the most recent file at the top of the list. To use a checkpoint file:

- Select the file to run by clicking on its row.
- Optionally delete some or all files using the **DELETE >** option.
- Optionally reduce the "**#commands to execute**" to a smaller value, perhaps to omit the last command(s) which caused a crash.
- Click on **APPLY** to run the file.

In this example the user has selected the oldest file and is about to delete the remainder using the **DELETE > ALL_BUT_SELECTED** option.

If you want to ignore all checkpoint files and run interactively just **DISMISS** this panel.

2.7.2 What happens when a checkpoint file runs.

Each command in the file is repeated verbatim, as if you had typed or mouse clicked it, until either the end of the file or

the "#commands to execute" value are reached. Thereafter the session returns to being interactive in the normal way. A new checkpoint file is written which, initially, will be a copy of the one being played back, but will then contain any further interactive commands.

2.7.3 Limitations of checkpoint files.

Although they are a powerful tool for recovering from crashes checkpoint files are not perfect. In particular they do not include any information about elapsed time between commands, which can lead to differences during playback in two situations:

- When D3PLOT animates each frame is displayed in the "dead" time between user commands. In effect the code says "Has the user given a command? No? Good, let's animate another frame while he is thinking." Animation will not actually commence during checkpoint file playback, even if **PLAY >** has been recorded, as there is no

  "dead" time between successive commands in which to execute it.
  Therefore if the session included animation the image which is on the screen may be different to that when the checkpoint file was recorded, and this may affect the outcome of any screen picking operations. Some more subtle consequences may also arise: for example contour bands may be different because the code has not yet decided to autoscale bands over all frames in an animation sequence.

- When the T/HIS <=> D3PLOT link is used this too may not play back correctly. The reason is that the two codes run independently and talk to one another via inter-process communication. Because checkpoint file playback leaves no intervals between successive commands, the remote programme (T/HIS) may not have had time to perform the operations requested, and return results, so the sequence of stored commands may "run ahead" of

  what is actually happening on the display and effectively give answers to questions that have yet to be asked. Therefore checkpoint file playback of all but the simplest "linked" sessions is likely to fail because of the asynchronous way in which the two codes are running.

We hope to address the issue of asynchronous behaviour in future releases, but for the time being these limitations apply.

2.7.4 Sending checkpoint files back to Oasys for debugging.

We hope that you won't experience crashes but, if you do, checkpoint files can help us to find and fix the problem as they answer the question "can you tell us what you did to make it crash?".

However environment variables, settings in your oa_pref file, and any "settings" files can all influence how D3PLOT runs; and we need these to replicate the status of the code when it crashed. So when you send us crash information please could you include as much of the following as possible:

- The checkpoint file(s) themselves.
- A copy of all your "**`oa_pref`**" files: in $OASYS, $HOME and $cwd. (See Appendix II for details)
- Any "settings" files (**`d3plot`*`nnn`*`.set`**)
- Any environment variables that have been set (eg **`MENU_AUTO_CONFIRM`**, see Appendix IV for a list of these)

2.7.5 Preventing the reading and writing of checkpoint files

When performing batch (non-interactive) post-processing, for example driven by FAST-TCF or PRESENTER, you are relying on running scripts of commands that assume a given programme state. These can be upset if checkpoint files exist on disk since they will have no mechanism for dealing with them.

Therefore the environment variable **`SUPPRESS_CHECKPOINT`** may be set, with the following result:

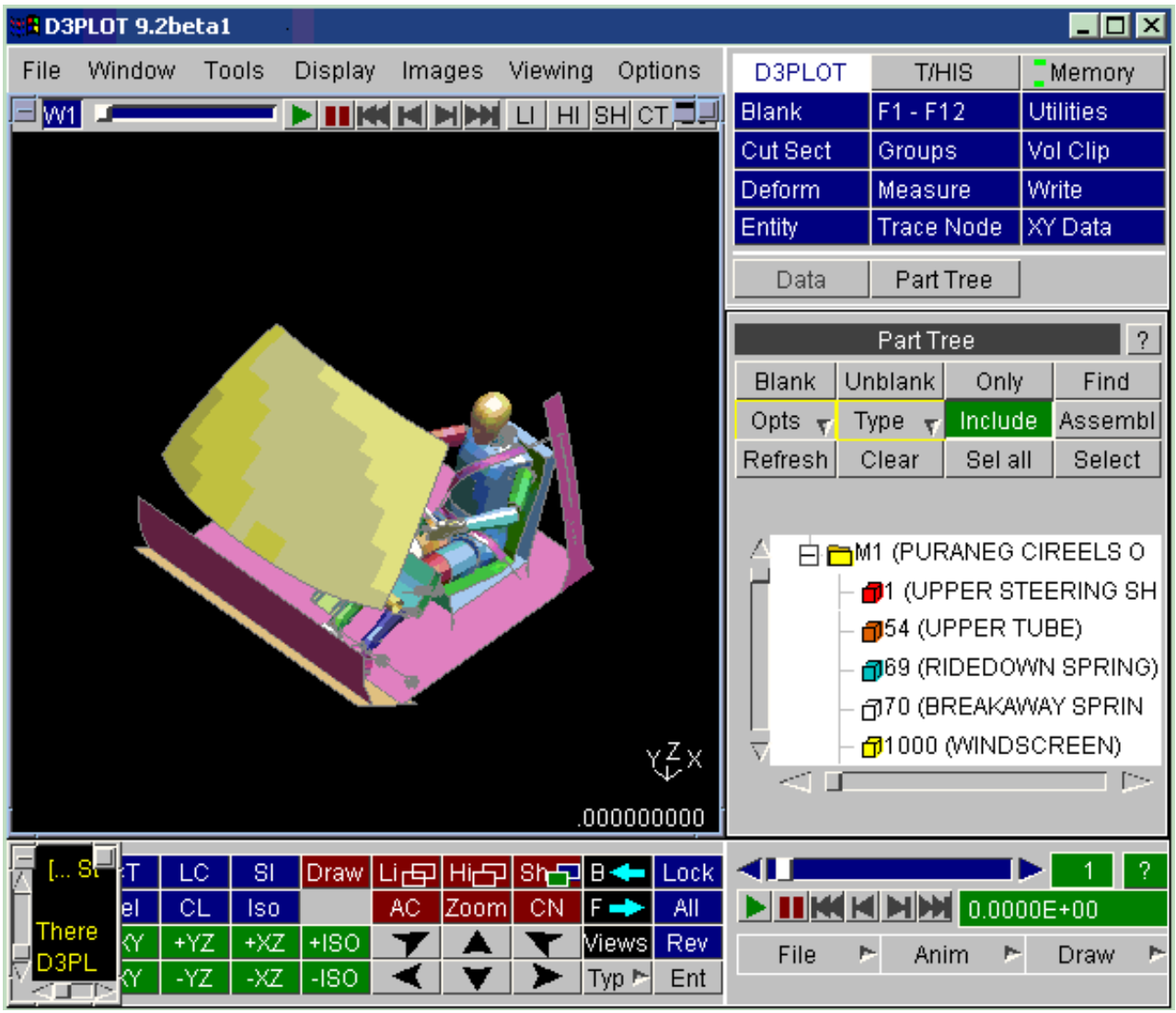
- Any existing checkpoint files will be ignored, and no questions asked about them
- No checkpoint file will be written during this session.

This and other environment variables are explained in Appendix IV.

3 USING THE D3PLOT SCREEN MENU SYSTEM

3.1 Basic screen menu layout

D3PLOT runs within a single window, owned by the window manager, which has several sub-windows inside it. A typical D3PLOT session will look like this:

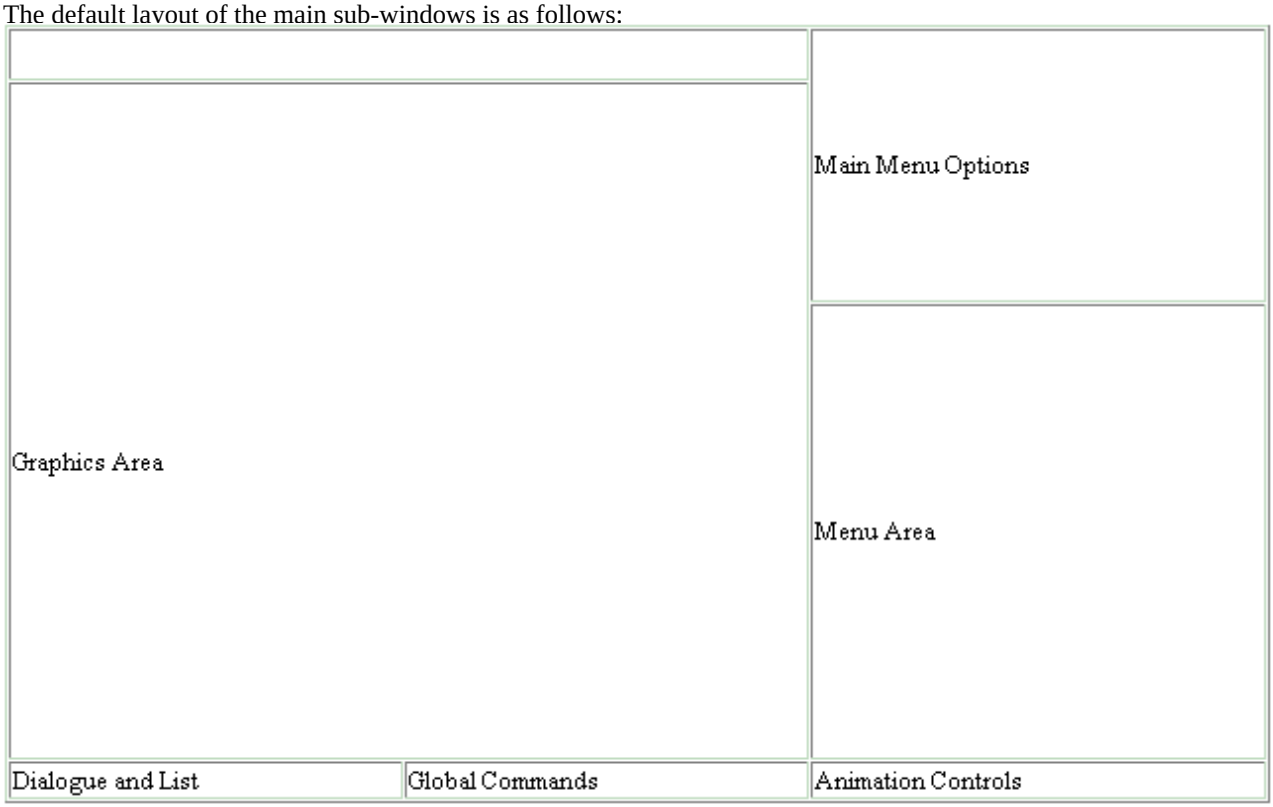


The various sub-windows always exist within the master window, and may be moved and resized at will inside it. They will keep their relative size and position as the master window is changed in size and/or shape, and will reappear after the main window is de-iconised.

Their exact location and size will depend on the size and resolution of the display: you can use the **DISPLAY_FACTOR** variable (see Section 2.2) to override default sizes and resolution.

The **TIDY** button in the icons box may be used at any time to restore this default layout: any unwanted sub-windows will be closed and the screen will be restored to the appearance here.

The default lavout of the main sub-windows is as follows:



Main Menu Options	Provides access to the majority of the commands and options available in D3PLOT through a series of sub menus
Graphics area	Is where graphics are drawn.
Dialogue & list	Allows "command-line" input and output, also provides a listing area for messages.
Menu Area	Displays the commands and options associated the current selection from the main menu options.
Global Commands	Gives access to commonly used commands
Animation Controls	Controls states and what is displayed during animation

While you are free to re-position these master windows it is recommended that you keep to this default layout. This is because when further sub-windows appear their position and size is designed assuming this layout, and aims to obscure as little useful information as possible.

3.2 Mouse and keyboard usage for screen-menu interface

All screen-menu operations are driven with the left mouse button, with the following exceptions:

Text in the dialogue area and text boxes requires keyboard entry;

Text strings saved in the cursor "cut" buffer may be "pasted" into dialogue areas and text boxes using the middle mouse button.

Popup" menus are invoked using the right mouse button.

The primitive "widgets" in the menu interface are used as follows:

BUTTONS:

Screen buttons are depressed by clicking on them, but action only takes place when the mouse button is released, so it is safe to drag the (depressed) mouse around the screen.

Buttons may be set (ie depressed) by D3PLOT itself, for example the **Solids & Shells** one above, to indicate that this option is in force. They may also be greyed out, for example the **Cont Surfs** one above, to indicate that the option is not currently available. Some buttons repeat automatically when held depressed: this depends on context. Buttons with "**...**" after them will invoke sub-menus.

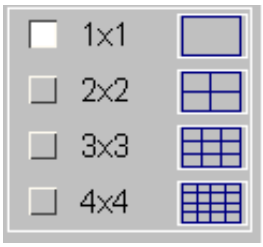
SLIDERS:

Sliders are moved by clicking on the slider button itself, and then dragging it to a new position. They may also be moved automatically by clicking on, and holding down, one of the arrows at either end.

TEXT BOXES:

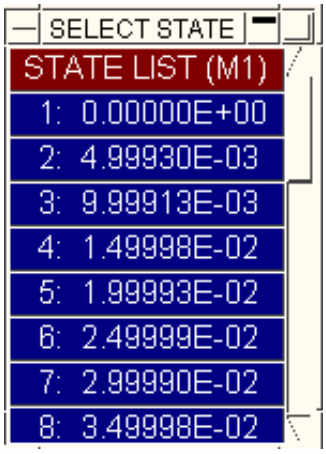
To enter text in a text box: first make it "live" by clicking on it, then type in text, then type **<return>** to enter the string. Clicking on a "live" box for a second time is exactly the same as typing **<return>**, so clicking twice on a box effectively enters its current contents. You can use the left and right arrow keys for line editing within a box: text entry takes place after the current cursor position.

RADIO BOXES:



A "radio" set is provided where only one selection is possible from a range of options. In this example the laser postscript output has been set to a single image per page. To select click anywhere on the row of the relevant option, any previously selected item will be deselected.

MENU LISTS:



Menus of items are used when you need to make one or more selections from a (potentially) long list. Click on the row you want to select: clicking on a row that is already selected will have the effect of unselecting it. When the list is too long to display in the window you can use the vertical scroll-bars to move up and down it.

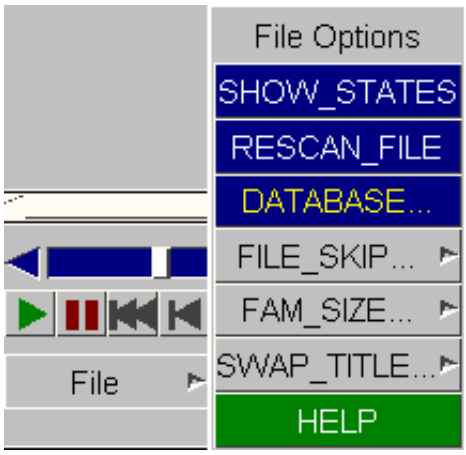
POPUP MENUS:

Where a button has a "right arrow" **>** symbol it means that a popup menu is available.

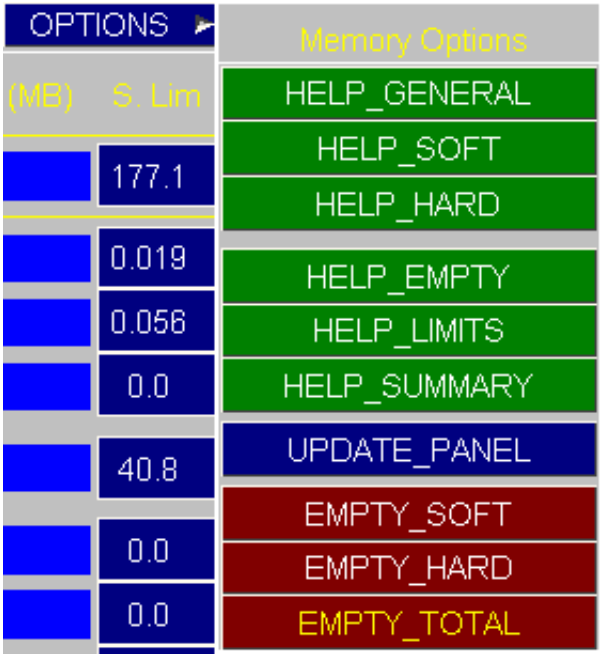
Click the right mouse button and the menu will appear. Holding down the right mouse button drag it onto the item you want.

Popup menus can be nested to any depth.

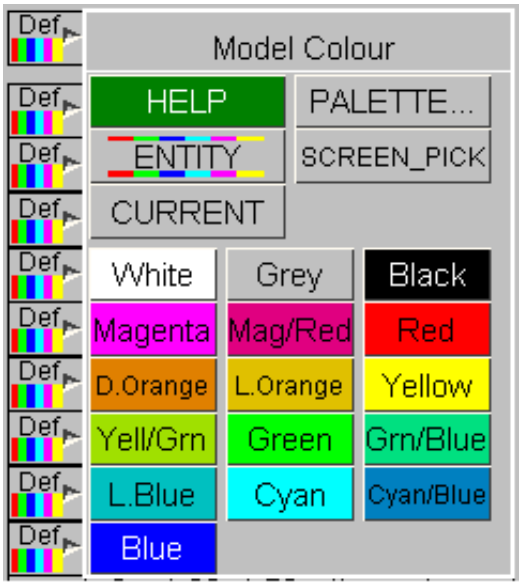
Note that popup menus can be invoked both from "clickable" buttons and from"non-clickable" ones: it makes no difference to their functionality.



Popup menu invoked from "clickable" **OPTIONS>** button



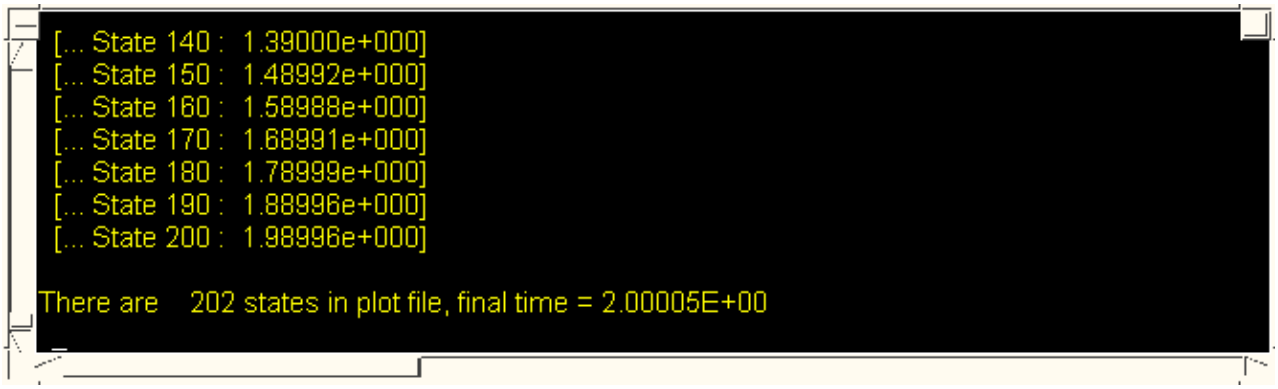
Popup menu invoked from "non-clickable" **DEF>** (colours) button



3.3 Dialogue input in the screen menu interface

The full command-line capability is preserved when D3PLOT is running in screen-menu mode, and you are free to mix command-line and mouse-driven input at will. There are some situations in which command-line input is more efficient: for example when entering lists of explicit entities.

Commands are entered in the dialogue box:



As this example shows the dialogue box is also used for listing messages, warnings and errors to the screen. It can be scrolled back and forth (its buffer is 200 lines long) to review earlier messages. The following colours are used:

Normal messages and prompts	Yellow
Text typed in by you	White
Warning messages	Magenta
Error messages	Red

There is a minor limitation when mixing command-line and screen-menu mode: you cannot perform the same function simultaneously in both modes. If you attempt to do so you will get the message:

`WARNING: recursive access attempted`

And you will not be permitted to continue. To clear this situation either close down the menu-based operation, or return to the main menu ("**`/`**" command) in the dialogue box.

3.4 Window management in the screen interface

Moving, resizing and scrolling of windows is based on the conventions used in the Motif Window Manager.

To move a window:

Click down on its title bar, then drag the window to where you want it to be. A "rubber-band" outline moves to show the window's current position. Where a window does not have a top title bar click anywhere on its grey background and drag it.

To resize a window:

Either Click on a border bar to move just that side, or on a corner bar to move both sides attached to that corner. Again, a rubber-band outline shows you the new shape.

or Use the **MAXIMISE** button in the top right hand corner of the window to increase the size of the window to the largest required size.

To scroll a window:

If a window has got too small for its contents then horizontal and/or vertical scrollbars will appear. Click on a scrollbar slider and move it to the desired position, the window contents will scroll as you do so. Alternatively click on the arrows at either end of the scrollbar for timed motion in that direction.

To iconise a window:

Click on the **ICONISE** button in the top right hand corner of the window.

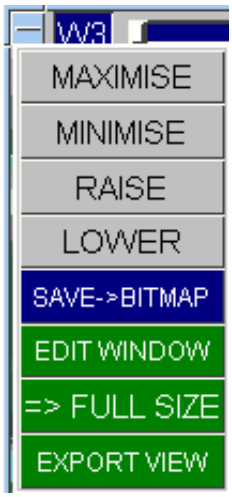
To restore a window:

Iconised windows may be restored by clicking on the icon in the **ICON** area.

Further options:

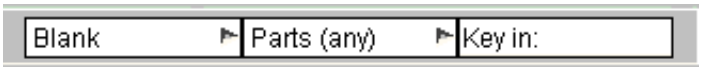
Click on the **OPTIONS** button to get the drop-down menu of window management options:

RESTORE	Restores an iconised window
ICONISE	Iconises this window (see below for iconisation)
RAISE	Raises this window to the top of the stacking order.
LOWER	Lowers this window to the bottom of the stacking order.
SAVE->BITMAP	Generates a windows bitmap (**`.bmp`**) file of the sub-window. This is an uncompressed file with a depth matching the number of bit-planes of the window. (This often doesn't work for the graphics window, since it uses mixed "visuals": use the **IMAGE** option instead for this.)



3.5 "QUICK PICK" Options.

Across the top of the graphics area is a block of 2 buttons. These are referred to as the **QUICK PICK ENTITY** button and the **QUICK PICK OPERATION** button in the following section.



By default the **ENTITY** and **OPERATION** buttons are set to **PART** and **BLANK.**

Possible operation types are:

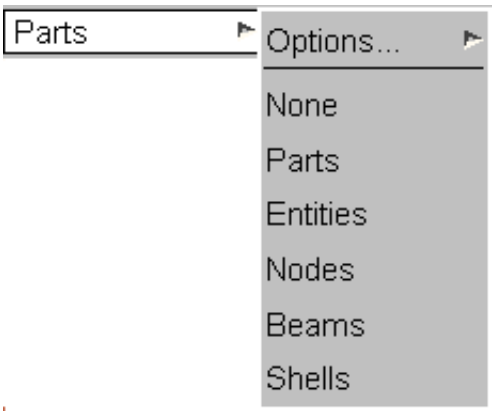
BLANK (default)	Section 6.1
UNBLANK	Section 6.1
ONLY	Section 6.1
COLOUR	Section 4.2.2.2
TRANSPARENCY	Section 4.2.2.4
DISPLAY MODE	Section 4.2.2.2
LABEL	Section 6.4
OVERLAY MODE	Section 4.2.2.2
OVERLAY COLOUR	Section 4.2.2.2
BRIGHTNESS	Section 4.2.3
SHININESS	Section 4.2.3
LOCATE IN TREE	Section 10
PROPERTIES	Section 4.2.2.3

Whenever these buttons are visible in a graphics window "quick picking" is active, and the cursor is live. Mouse buttons have the following functions:

Left	"Do" the operation. For example blank the entity if in BLANK mode, change its colour if in COLOUR mode, etc. Drag across the screeen using the left mouse button to selecet multiple entities by area.
Middle	"Undo" the most recent operation. Thus unblank the last pick, etc.
Right	Raise the full options menu for the selected object type, giving the option of performing any of the "quick" operations on it, regardless of the current mode.

In all cases the effect is immediate, for example clicking on a part to blank it results in the image being redrawn.

The **ENTITY** button can be used to access a popup via the right mouse button to change the default selection category from **PART** to any of the generic element classes that the model contains (PARTS, NODES, SOLIDS, SHELLS, THICK SHELLS BEAMS, SPRINGS etc).

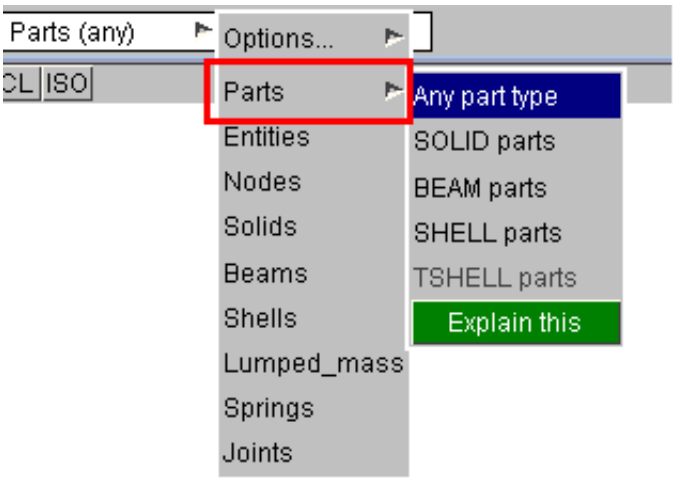


PARTs are a special case:

The default setting for PART picking is to pick parts of any eligible element type.

But because the default screen picking process will tend to pick 2D and 3D elements (because they have a finite area), it can be difficult to pick **BEAM** parts if they overlay 2D or 3D mesh.

Therefore it is possible to restrict the type of part to be picked by underlying element type.



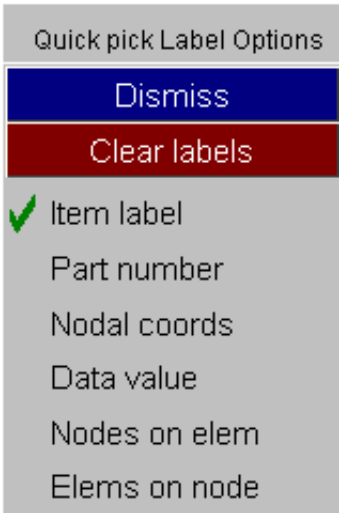
In addition to being able to **BLANK** items the **OPERATION** menu can be used to select the operations listed above to apply to the items selected.

The **OPTION** button can then be used to select the colour, transparency level, display mode etc that is applied to the item when it is selected on the screen.

(Not all options will be available for parent item types.)

All operations carried out using these options and the mouse are stored and can be undone using the **MIDDLE** mouse button. Furthermore all of these options can be used while animating. To indicate which operation is currently active the mouse symbol will change as is appropriate.

Labelling

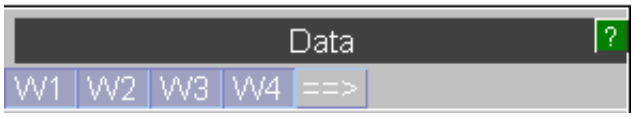


Any permutation of of the following items can be selected for labelling, but note that some can only apply to nodes and some to elements.

Item Label	Draws the node or element label
Part Number	Draws the number of the part the item belongs to
Nodal coords	Draws the current [X,Y,Z]coorrdinates to the left of the node
Data value	Draws the data value associated with the currently visible plot data(eg CT,SI) to the left of the node position or element centre. A value of 0.0 is used if the currently visible plot does not imply data (eg LI,HI,SH drawing modes) or if there is an entity type/data plotting mismatch (eg beam element data froma contact surface data plot).
Nodes on elem	Draws the labels of all nodes on the selected element
Elems on node	Draws the labels of all elements attached to the selected node

The labels persist during animation and redrawing, until the the **Clear Labels** option is selected.

3.6 "Tabs" for multiple graphics windows.



When more than one graphics window is in use most menu panels will have a "tab" button for each graphics window: **W1**, **W2**, etc. In this example there are four graphics windows.

These tabs control the graphics windows to which the commands issued in this panel apply: here any command would apply to all four windows.

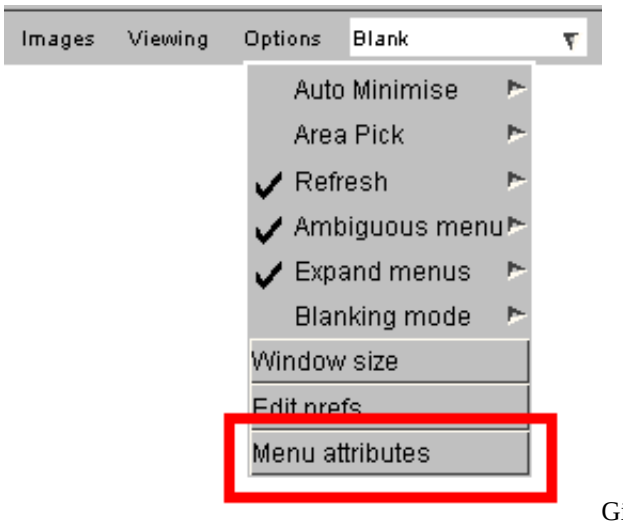
The **==>** button takes the settings of the first active window in this panel and copies them to all other active windows. In this example the view current in Window #1 would be propagated to windows #2 to #4.

Multiple graphics windows are discussed in more detail in section 2.6.

3.7 Customising the User Interface

3.7.1 Customising Menu size, fonts, colour and handedness

As mentioned in section 2.2 the scale of the menu interface, the font typeface and size, and also the left-handedness of the menu interface may be customised interactively using **Options > Menu Attributes.**



Gives the menu attributes panel:

Display Factor	Is a factor on the overall scale of the display, lying in the range 0.5 to 2.0, default 1.0. Larger values make the display seem bigger to the software, resulting in smaller menu panels and fonts. Smaller values increase the size of menu panels, buttons and fonts, and can be useful for the visually impaired. This factor can be especially useful on "wide screen" displays with very asymmetric horizontal and vertical resolutions. The operating system *should* determine the physical size of the display correctly. However we have observed a few instances where this does not happen, the symptoms being that fonts and menus appear either far too big or too small and cannot be corrected by using **Display Factor**. In this situation you may need to tell the Oasys software the physical dimensions of your display, and this process is described under "Setting the correct physical resolution for your display" in section 3.2 of the extra section on graphics in the Primer manual.
Font size	Controls the size of fonts used in the menu interface (but not for graphics). This works independently of the Display Factor, allowing further fine-tuning of the appearance of the user interface.
Font Typeface	For most applications the default Helvetica (Arial on Windows) will suffice. But you can also choose Times or Courier, and Bold variants of all of these.
Brightness Saturation	These affect the overall brightness and also the colour saturation of the user interface. They both lie in the range 0.0 to 1.0, default 1.0.
Left-Handed support	By default D3PLOT is set up for right-handed usage, which has influence on both mouse buttons and the keyboard "meta" keys: <shift> and <ctrl>. (The left and right meta keys have different functions during dynamic viewing: see dynamic viewing) You can swap the handedness of mouse and/or meta keys, which will reverse them in the left <=> right sense. **Note**: This swapping is local to D3PLOT, and is applied after any system user interface configuration. So if you configure your computer to swap mouse buttons globally, then swap them here, the net effect will be to have unswapped buttons again!

Saving Menu Attributes settings

The attributes above may be saved in the "oa_pref" file by using **Save_Settings**. Subsequent sessions of D3PLOT will pick these up and re-apply them. The "oa_pref" file is described in more detail in Appendix II.

For backwards compatibility these attributes may also be set using environment variables as described in Appendix XIII. Where conflicting settings exist those in the "oa_pref" file generated by the panel above (or by hand) will "win".

Note: Oasys software potentially reads three "oa_pref" files when an application starts, in the following order:

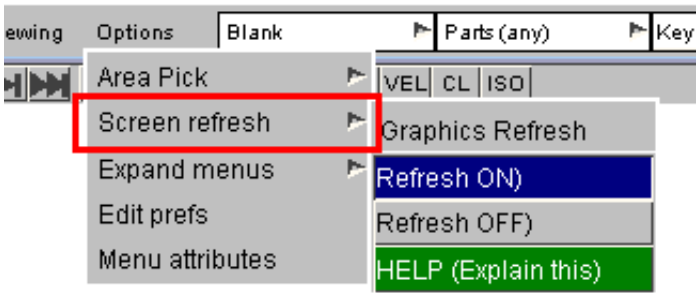
(1) The $OASYS directory
(2) Your "home" directory ($HOME on Unix/Linux, $USERPROFILE on Windows - typically c:\Documents and Settings*user_id*)
(3) The current directory (typically "Start in" directory on windows)

Save_Settings in this panel update the file (#2) above in $HOME / $USERPROFILE, on the principle that you will have write permission there and - usually - it will not affect other users. However all "oa_pref" file settings are applied on the "last found wins" basis, so if you have file in your current directory with different settings these, being the last to be found, will "win".

3.7.2 Screen Refresh: Controlling graphics window redraws

Normally graphics images in D3PLOT redraw at an acceptable speed, and the delay when refreshing "holes" left by menus popped up in front of the graphics window are not objectionable.

However if you are running a large model on a slow machine this may become a problem, and it is possible to turn off screen refreshes using **Options > Screen Refresh**. This will leave black holes when menu panels are unmapped, and you will have to issue an explicit redraw command to get rid of these. You can turn the refresh switch on again at any time.



You can save the graphics refresh status in the "oa_pref" file with the line:

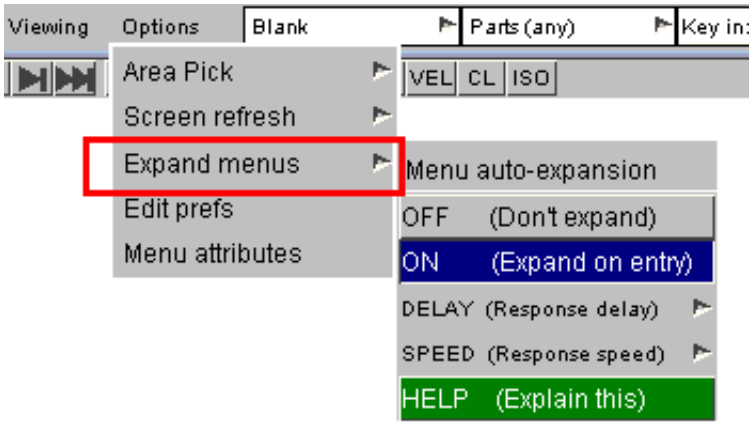
```
primer*graphics_refresh:  off |
                          on
```

The default is "On".

Note: Unlike Oasys PRIMER no "backing store" drawing is used in D3PLOT, so issues such as Bitmaps, Pixmaps and PBuffers do not arise.

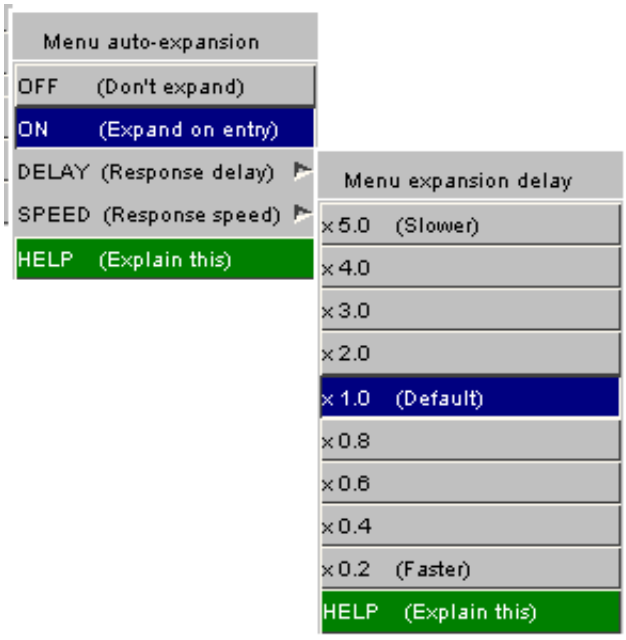
3.7.3 Menu "Auto Expansion"

A few of the menus in D3PLOT are too narrow when first mapped to show all the columns of their data, so by default "auto expansion" is enabled. This causes the menus to widen themselves, typically to 90% of the enclosing width available, after a brief delay. You can control this behaviour using **Options > Expand Menus** as follows:



By default manu auto-expansion is **ON**, but you can suppress it by turning it **OFF**.

Controlling the speed and delay:



You can also control:

DELAY the time interval between the mouse entering a window, and the window starting to expand.

The delay time is controlled as a factor on the default behaviour.

The actual delay time will vary from system to system depending upon the Window system and underlying speed, but a typical delay will be approximately 0.5 seconds.

SPEED (Not shown here) is the rate at which the menu expands and contracts.

As above it is controlled as a factor on the default speed.

Saving Menu Auto Expansion Settings

The menu expansion parameters may be saved for future PRIMER sessions by setting the "oa_pref" file options:

```
d3plot*menu_expand:        ON | OFF
d3plot*menu_expand_delay:  Floating value in the range 0.1 ... 5.0
d3plot*menu_expand_speed:  Floating value in the range 0.1 ... 5.0
```

Full details of all "oa_pref" file options and environment variables are given in Appendix II

3.8 Shortcut Keys

Some panels and actions have shortcut keys assigned to them:

Plotting Modes

H - Hidden mode plot

S - Shaded mode plot

L - Line mode plot

F - "Fringe" (SI plot)

View Controls

A - Autoscale current image

V - View control panel

Z - Zoom using cursor

"+" - Zoom in (factor 2.0)

"-" - Zoom out (factor 0.5)

1 - +XY view

2 - +YZ view

3 - +XZ view

4 - +ISO view

5 - -XY view

6 - -YZ view

7 - -XZ view

8 - -ISO view

0 - "Exports" the view of the current graphics window to all other active windows

Blanking

B - Blanking control panel

R - Reverse blanking of image

U - Unblank all

Panels

C - Close all panels

D - Drag cut plane

E - Entity panel

M - Measure panel (node -> node)

O - Overlay and Display panel

P - Properties panel

W - Write image file panel

X - Cut sections panel

N - Pick cut plane node(s)

State Selection

-> - Forward one state

<SHIFT> + -> - Forward one frame

<- - Backward one state

<SHIFT> + <- - Backward one frame

<HOME> - Jump to first state

<SPACE> - Toggle animation

<END> - Jump to last state

Windows

G - Open new window

I - Iconise windows

T - Tidy all windows

<DEL> - In a graphics window erases dynamic labels

Miscellaneous

Q - Quit current pick action

<PAGE DOWN> - Move down a page

<PAGE UP> - Move up a page

If the mouse is in a graphics window, commands that imply a graphical change apply to that window only; otherwise they apply to all active graphic windows.

4 BASIC DATA EXTRACTION AND PLOTTING

This section describes how to extract data from disk, and how to display them graphically. Screen menu usage is assumed, although brief references will be made to equivalent command-line instructions where appropriate.

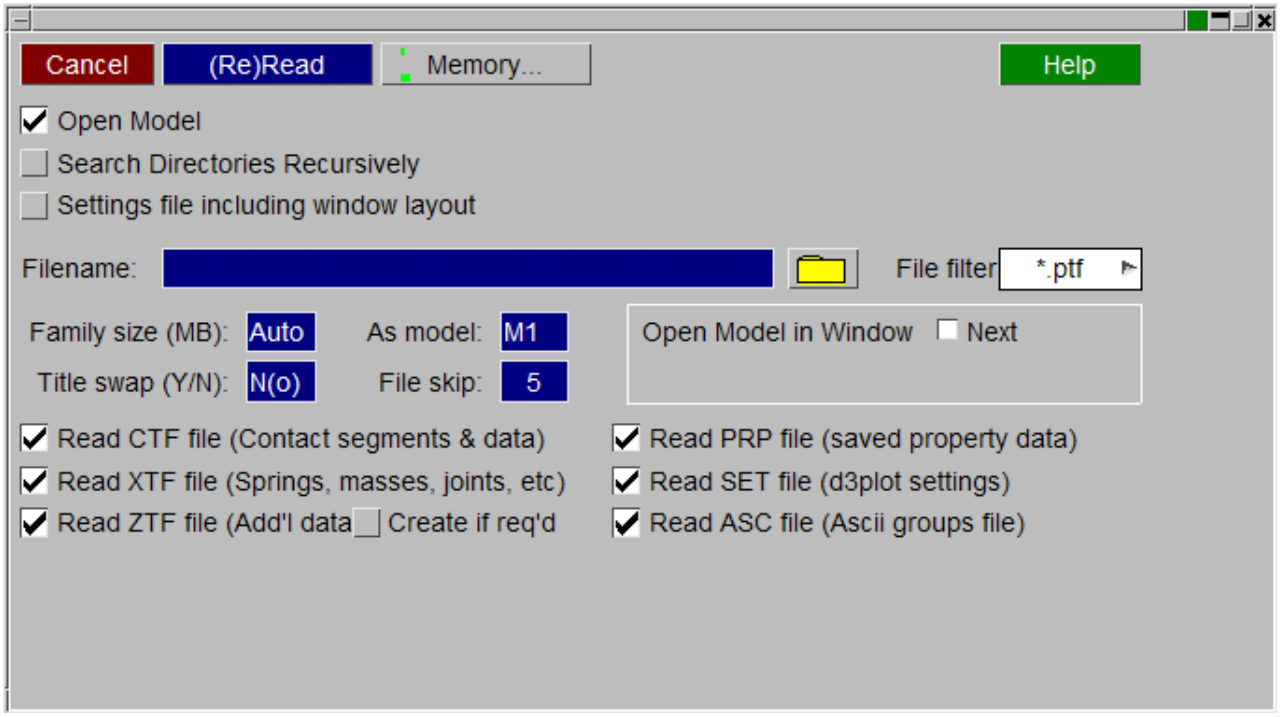
4.1 Reading results from an analysis.

4.1.1 Reading plot and other files at initial programme startup

The standard filename input panel is:

By default this panel will allow you to select a single model and open it.

Alternatively this panel can be used to either search directories for results and open open multiple models (see Section 4.1.2) or a setting file containing model information can be loaded (see Section 4.1.3).



You can type a filename into the text entry box (after **File:**), or use the button to obtain a standard file filter box .

The **File filter** button controls what extension will be used to search for file types in the file filter box. See below.

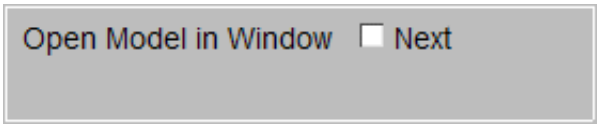
`*.ptf`	Finds normal complete state files
`*.rlf`	Finds dynamic relaxation files
`d3*`	Finds unnamed default files (d3plot, d3drlf, etc)
`*`	Finds all files

QUIT aborts the file opening operation.

REREAD allows you to re-open the current file.

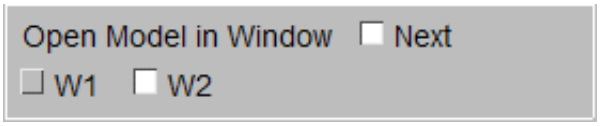
Open Model in Window:

By Default D3PLOT will open each model in a new Graphics Window.



If a second or subsequent model is opened D3PLOT will offer the choice of opening the model in a new Window (**Next**) or one of the existing Windows (**W1**, **W2** ...).

If more than one Window is selected then D3PLOT will add the model to each of the Windows.



Family size (MB):

The binary output files written by LS-DYNA form "families". Each family has a root member and may have children. The maximum size of any member of a family is set when LS-DYNA runs, the default being 7MB, however any size > 1MB can be used.

By default D3PLOT determines the family member size automatically (the **Auto** setting shown here), but you can override this by entering a size in MB. This is almost never necessary: read Section 9.1 before doing this.

File skip:

File families should form a contiguous sequence (root, member #1, member #2, ...) But it is sometimes the case that members are missing: intermediate members be deleted to save disk space, and occasionally LS-DYNA skips members.

The **File skip** value (here zero) is the number of missing members that D3PLOT will skip before giving up its search and deciding that it has found the end of a file family. See Section 9.1.3 for further information.

Title swap:

It is unfortunately the case that some versions of LS-DYNA have been compiled with numeric conversion flags which endian swap their output. This works fine for numbers, but scrambles the title (the string ABCDEFGH becomes DCBAHGFE).

If your title looks like garbage try changing this field to **Y(es)** to see if this fixes the problem. (Note that you can do this at any time during a D3PLOT session: see Section 4.1.7).

MEMORY...

Maps the standard memory management box (see Section 11.7). This allows you to set database memory limits and display mode before opening the file, which may be necessary in exceptional circumstances.

Read CTF, XTF and ZTF files

By default these three files are read if they exist, but you may suppress the reading of any if you wish. In a model with many contact surfaces omitting the **CTF** file can speed up graphics and save memory, since contact segments will be ignored.

Omitting the **XTF** file contents will not give similar savings, and is not generally recommended. MPP versions of LS-DYNA do not write a **XTF** file, so from V9.0 onwards the equivalent data is also present in the ZTF file.

It is recommended that both **XTF** and **ZTF** files should always be read if present.

Details of the contents of all these files are given in section 9.2

Read PRP, SET and ASC files:

PRP (Property) files, ***<filename>*_nnn.prp**, contain model-related information such as colour, transparency, overlay, etc that has previously been written from the **PROPS** panel. These files will be reread if found so that all this status information is automatically restored.

SET (Settings) files, **d3plotnnn.set**, contain information about programme settings on a per-window basis, such as background colour, cut-sections, data component, etc. These will be read if found and will automatically create new windows if required, and set them up as they were before.

Settings and Properties files are described more fully under **UTILITIES, SETTINGS_FILE** .

ASC (Ascii groups) files, **group001.asc**, contain optional group information in a human-readable form which matches that used by Primer in the keyword input deck. If such a file is read it will supersede those groups currently stored for this model.

Group handling in release 9.0 of D3PLOT has been extensively modified and improved, and it is described more fully under **GROUPS**.

File filter (Maps the standard file selection filter box -

D3PLOT requires a complete state, or equivalent, file in order to run. Under Oasys conventions this will have the filename format **<name>.ptf**, but any name is acceptable so long as the contents are recognisable.

The default filter box "pattern" is set to ***.ptf**. Some other naming conventions you may encounter are given below. (See Section 4.1.12 for a list of supported file types and names.)

d3plot	Default complete state filename chosen by LS-DYNA if no user-defined name is given (ie G=xxx is missing from the input line).
d3drlf	Default dynamic relaxation filename chosen by LS-DYNA if no user-defined name is given (ie B=xxx is missing from the input line).
<name>.rlf	Dynamic relaxation output file using normal naming conventions.
d3eigv	Default eigenvalue analysis filename from LS-NIKE.
n3plot	Default transient analysis filename from LS-NIKE.

To make using the standard file filter box easier you can pre-select the "pattern" that will be used for scanning files on disk using the options shown here. Of course any pattern can still be typed into the file filter box itself.

Adaptively remeshed analysis filenames

D3PLOT supports LS-DYNA adaptive remeshing, in which a series of families are generated with a mesh that is progressively refined (see Section 4.1.11).

If the "base" **<name>.ptf** (or d3plot) file is selected then all successive familes (**.ptfaa**, **.ptfab**, etc) are read in and their states are concatenated internally.

To read in a given family only select is base member (eg **<name>.ptfad**) explicitly and only that remesh family will be read.

Eigenvalue (modal analysis) files.

D3PLOT supports output files from modal analyses. They are treated in exactly the same way as transient analyses except that:

- Each "state" is a modeshape, and the States Slider moves between these.
- Animation works on the currently selected state only, oscillating it through +/- 180 degrees.

Domain Decomposition files from MPP analyses

The MPP version of LS-DYNA can write a pseudo PTF file that shows the domains into which a model has been decomposed for parallel analysis. It contains undeformed geometry only.

This is not a true PTF file and, in particular, each domain is a part that contains elements of all types in its region. Strictly this is illegal: parts can only contain elements of one type, so D3PLOT handles this automatically as follows:

- The special file type is diagnosed automatically from its contents
- The composite parts of each domain are split into separate solid, beam, shell and thick shell element parts
- These are given labels that make them the same colour in D3PLOT's default colouring scheme
- Groups of each domain, containing all parts in that domain, are automatically constructed.

In this way the technically illegal domain decomposition file can be processed normally, and the use of groups makes the sketching and (un)blanking of domains very straightforward.

Echo of file contents.

Once you have given a filename its contents are scanned and checked for integrity. If an **.XTF** and/or **.CTF** file is found (and flagged for input) it too is scanned for any extra information it may contain.

You are given a summary of the file contents.

Assuming that these are acceptable click on **OK** to continue with reading in all the basic data from the file. If all goes well the basic nodal coordinates and element topology are read in, and you can start plotting. Various things can go wrong, however:

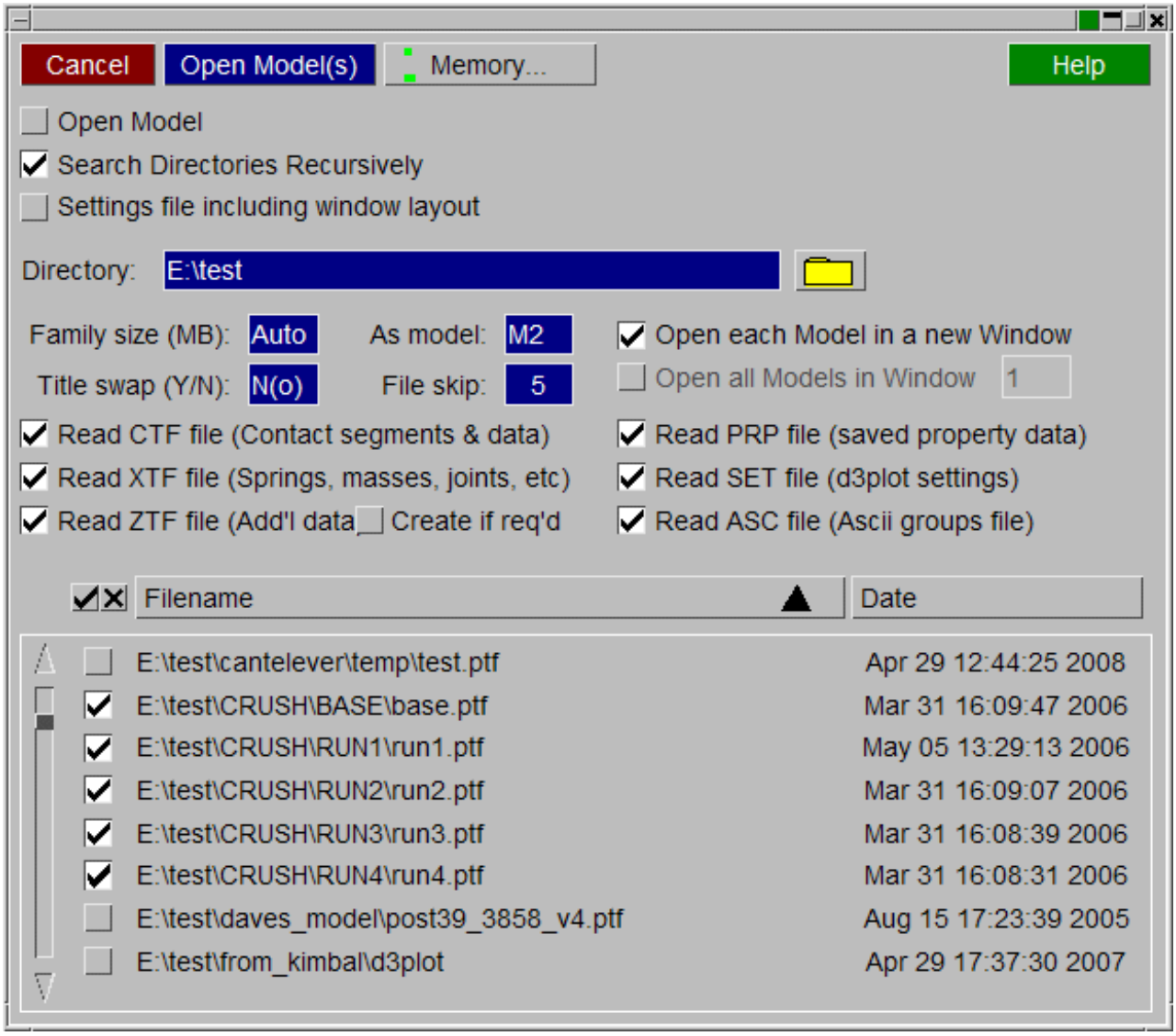
The title is garbled	Use the **SWAP_TITLE** button to see if this will fix it. If it doesn't then the file is probably the wrong format, corrupt, or not a complete state file.
The files are corrupt	If the job crashed, or is still running and has not finished its initialisation phase, the files may be incomplete and/or full of garbage. D3PLOT tries to check for this by checking file integrity, and it may reject the files if they are found to be too corrupt to handle.
The files are incomplete	Sometimes you will get the message that "**arbitrary numbering tables are corrupt, sequential numbering will be used throughout**." This is not fatal: the node and element arbitrary numbering tables are the last things written during the initialisation phase, and may not yet have been written from system buffers to disk if the job has just started. You can plot the undeformed geometry, but node & element labels will be shown in internal sequential order.
Binary format mismatch	The exact format of older files cannot be determined automatically, for example Cray and IEEE 64 bit formats can get confused. This can be fixed with the **FILE_TYPE_64** and **_32** environment variables. See Section 9.1 for more details.
PTF/CTF/XTF mismatch	If the contents of the **.CTF** and/or **.XTF** files do not match those of the **.PTF** file, (ie they are leftovers from an earlier analysis), you are warned and their contents are rejected. The plot (**.PTF**) file is always taken to be the truth.
ZTF file mismatch	Because the **ZTF** file is generated from the input deck (**.key**) it may not match exactly the other output files. This is tolerated: un-matched items are simply ignored.

4.1.2 Search Directories Recursively

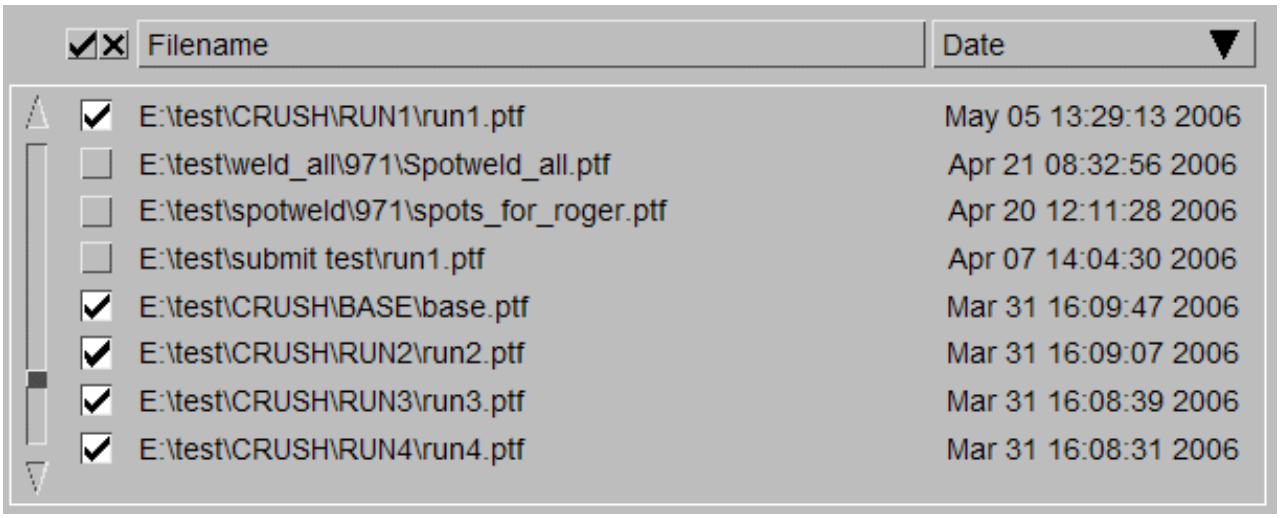
Multiple models can be opened by using the option to search directories recusively.

After a directory has been specified D3PLOT will display a list of all the models it can find in the directory structure and each file can be selected

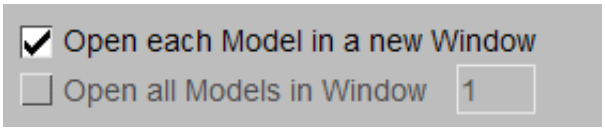
If the user selects more than 32 models or if the number selected + any models already loaded into D3PLOT is greater than 32 then D3PLOT will open the models selected until the limit of 32 is reached.



The list of models can be sorted by either alphabetically by directory name or by date into either ascending or decending order.



When multiple models are read each model can either be read into a separate Window or all of the models can be loaded into an existing Window.

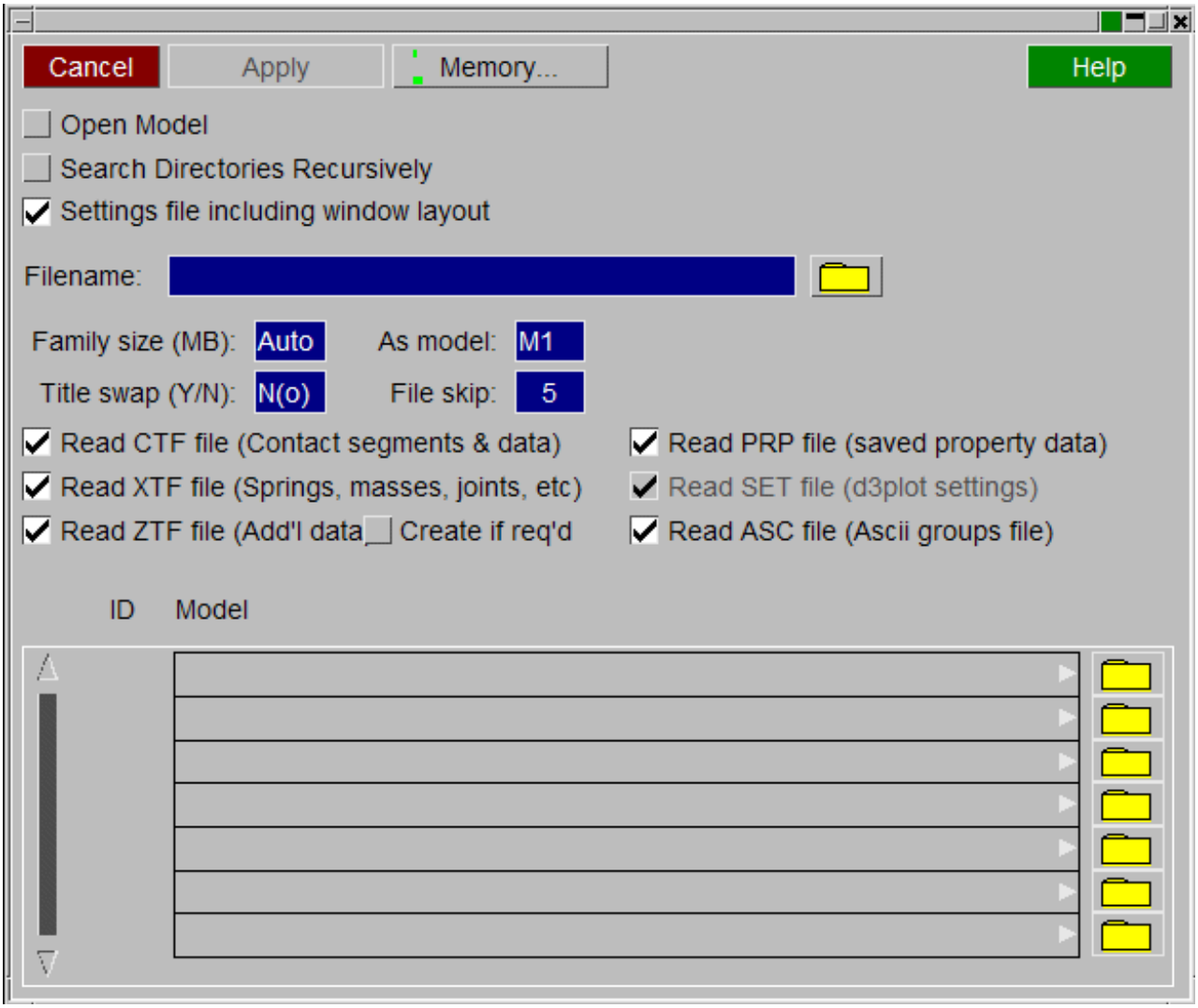


4.1.3 Settings File Including Window Layout

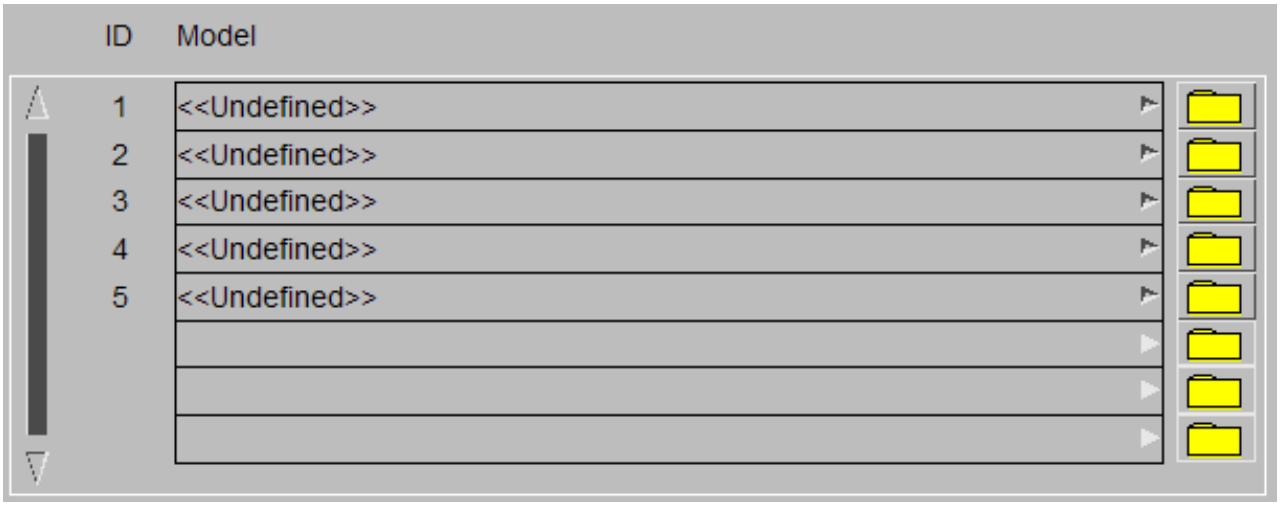
From version 9.3 onwards a D3PLOT settings file contains information on the number of models that were open when the file was saved along with the window and page layout. The filenames for the models are not stored in the settings file, just the number of models and which model (M1, M2 ...) was located in each window.

If the setting file was saved while the D3PLOT->T/HIS link (see Section 6.12) was running then the setting file will also contain information of the number of T/HIS graphs and it will contain a T/HIS FAST-TCF script which will regenerate the graphs contents.

This option can be used to reload a version 9.3 settings file and to restore the model and Window layout.

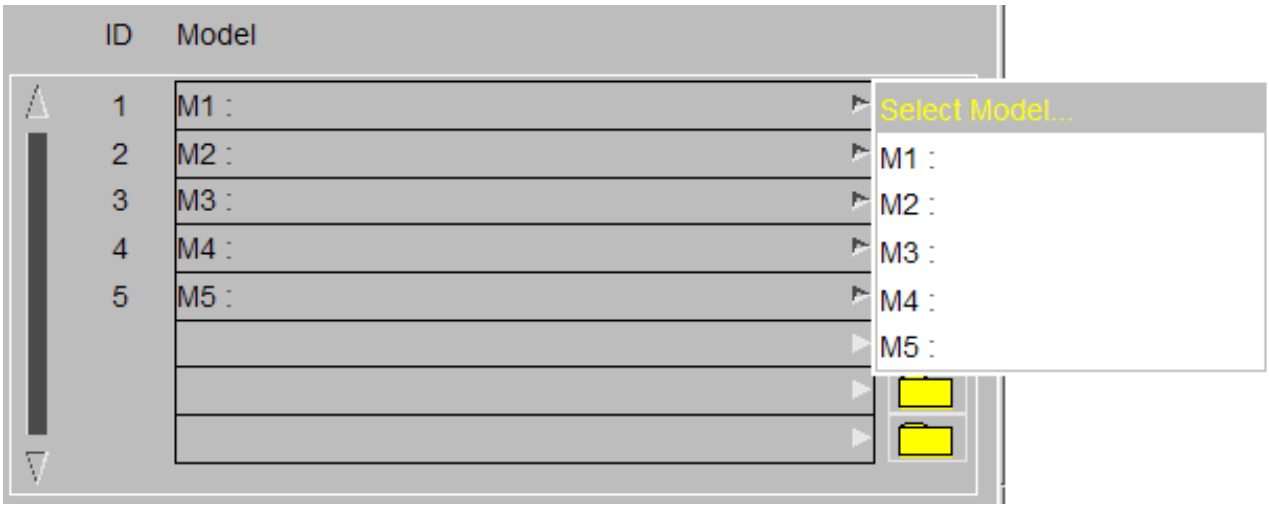


After a settings file is selected it's contents will be scanned to see how many models are required and D3PLOT will then display a list to allow the models to be selected.

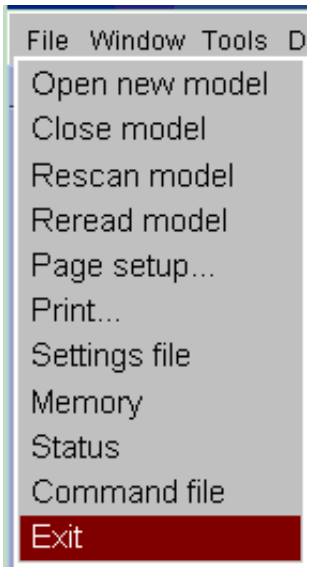


If models have already been read into D3PLOT they will be automatically selected for the models to use when replaying the settings file.

Any of the pre-selected models can be changed by using the popup menu to select a different model.



4.1.4 **Open New Model** Opening a new model file



From version 9.3 onwards D3PLOT supports up to 32 concurrent models.

You can open a new model file at any time using the **FILE>** popup which is located at the top left of the top options (**Main Menu**) box, **OPEN NEW MODEL** command.

This creates a new graphics window to contain the model, then reads a new filename in exactly the same way as described in Section 4.1.1 above.

If this is a second or subsequent model its window will inherit those attributes of the first window that can legitimately be transferred to this new one: background colour, data component, cut-sections, etc. Where attributes cannot be transferred, for example a data component that doesn't exist in the new model, then the programme defaults will be used.

4.1.4.1 **Close Model** Closing an existing model.

You can close a model at any time using this command. The model will be removed from any window in which it appears, and if it was the only model in such a window then the window will also be deleted and any remaining windows renumbered downwards to close the window numbering gap.

(Note that in D3PLOT V92 models may reside in the database without being displayed in any window, however they will continue to consume memory and models should be deleted if unneeded to free memory for other purposes.)

4.1.4.2 **Rescan Model** Scanning a running analysis for more states.

If an analysis is still running you can scan the file family for any further states that may have been written since you opened the model (or last scanned it).

Because of the way that computers work it is possible that the most recent state written from a running analysis may not be completely debuffered to disk. It this case D3PLOT will usually detect that the state is incomplete and offer you options for dealing with this, however it is usually best to ignore such states as attempting to read corrupt or incomplete data can lead to problems.

4.1.4.3 **Reread Model** Closing and reopening a model.

The **Reread Model** command differs from **Rescan** in that it closes a model completely then reopens it again (in the same window(s) that it occupied previously). It is the equivalent of **Close Model** followed by **Open New Model**.

You should use this instead of a **Rescan** when:

- An analysis file that is currently open in D3PLOT has been rerun from scratch using the same filenames, and needs to be reread in its entirety.
- You have an adaptive remesh analysis running and you want to scan for further family members (**Rescan** will only look within the current family member).

4.1.5 Basic animation, the "current state", and selecting states.

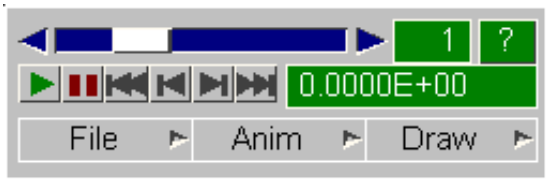
The programme maintains the concept of a "current state" for each window, which is that being displayed at the moment. When an animation is halted the state at which it stops becomes the current state, and any state selected explicitly by the user subsequently becomes the current state in its place.

Each graphics window is independent, and each may show different states. To provide both independent and collective control the following mechanisms are used:

The master **STATE NUMBER** slider applies to ***all*** windows for which its **W*n*** tabs are active.

Moving this slider to a new state will cause all these windows to jump to this state. **PLAY** makes all selected windows animate, and **STOP** halts them.

This range of this slider is 0 to the highest state in all models, and it can be the case that it permits selection of a state that doesn't exist in a given model. Selecting such a state is legal and leaves the window(s) of that model unchanged.



The local state slider and associated controls in the button bar at the top of each graphics window applies to this window ***only***.

Moving the slider, or using **<<, |<**, etc, lets you move between the animation states currently defined for this window.

By default all states in a model are selected for animation; but this can be limited to restricted states, or extra states can be displayed by interpolation.

Therefore these controls move between what has been selected for this window (or for modal analyses through the +/- 180 degree phase angle for the current modeshape).

All animation and static state selection is carried out in the **State Display** box. Its basic controls are described here, with more detail in the following sections.

To start animating

To initiate an animation simply press **PLAY** , and to halt it press **STOP** .

By default all states in the model will be animated at full speed in the current display mode. More information on animation is given in Section 4.4.

To select an explicit state by number

Either: Move the **STATE NUMBER** slider to the state you want, or use the arrows at its ends to scroll it left or right. When you release the mouse button the selected state becomes current and will be drawn.

Or: Type an explicit state number into the **State:** box. This will become the current state and will be drawn.

To select an explicit state by time

Type the required time into the **Time:** box. If a state with that time exists it will be used, otherwise the state with the closest time after that you have specified is used. This becomes the current state and is drawn.

The programme "knows" about all states in a file, they are scanned as part of the initialisation process, and it can jump directly to any state. Data required for plotting are read in selectively on as "as needed" basis.

States at interpolated times

The current (static) state shown in this box cannot be at an interpolated time between two explicit states: defining an intermediate time will result in the next highest state being used. However the sliders and state manipulation controls in each window can be used to move between interpolated states.
For more information on interpolation see Section 4.4.2.1**.**

Selecting and animating mode-shapes.

Most analyses are transient, and each state will show successive times. Modal (eigenvalue) analyses are performed in the frequency domain, and each "state" is a different modeshape at some frequency.

D3PLOT operates in much the same way except that:
- The "States" slider now moves between modeshapes (state 1 = mode 1, etc)
- Animation is performed by oscillating a single state through +/- 180 degrees
- The slider at the top of each window moves through the +/- 180 deg cycle.

Using keyboard "short cut" keys to cycle through states

You can use the following keyboard keys to select states:

<-- and --> arrow keys step backwards and forwards respectively. They loop round when they reach the limits of their respective directions.

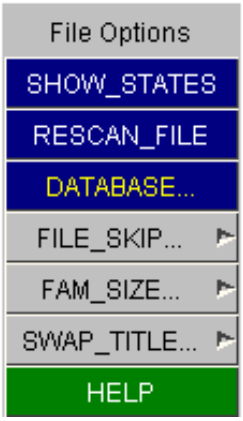
<Home> jumps to the first state
<End> jumps to the last state

As with all short cut keys the windows upon which they act are determined as follows:
- If the mouse is in a graphics window then they act only upon that window.
- If the mouse is in some other menu window then they act upon all active graphics windows.

To cycle through animation ***frames***, as opposed to states, use <shift> + left/right arrow keys. In most cases the only difference will be in data-bearing plots with contour levels set to "auto":
- Cycling through states will autoscale each image.
- Cycling through frames will use the envelope of contour values for the whole animation.

Click here for the next section

4.1.7 The **FILE >** popup menu options



You can manage many aspects of your database using the **FILE >** popup menu.

Some of these options can only apply to one model at a time. When this is the case the operation will be applied to the first active model as selected by the **W*n*** tabs in this panel.

SHOW_STATES - List all states in the file
If you think that you cannot see all the states that should be there you should consider the following possible reasons:
- Analysis is still running: You may need to **RESCAN_FILE** for newly created states.
- Missing family members: You may need to adjust the **FILE_SKIP** value.
- Wrong family member size: You may need to adjust the **FAM_SIZE** value.

RESCAN_FILE - Scan the file for any new states.

If your analysis is still running D3PLOT will not know about any states that may have been written since its initial scan of the file. **RESCAN_FILE** will search the file for new states and update the internal tables to show them.

DATABASE - Displaying and managing the results storage database D3PLOT loads results from file on as "as needed" basis, and may supersede unwanted results in memory to save space. The process is automatic and can normally be ignored, however users with big models may need to intervene to economise on memory usage.

FILE_SKIP - Jumping over gaps in family member sequence

Sometimes the family member sequence **<name>.ptf**, **<name>.ptf01**, ... **<name>.ptfnn** may contain gaps. This can be due to deliberate deletion of intermediate members to save disk space, or because LS-DYNA has skipped a member.

D3PLOT will skip over **<FILE_SKIP>** gaps before giving up its search for new members and deciding that it has reached the end of the file family. **<FILE_SKIP>** may be zero (no gaps) or any positive integer, but bear in mind that large values will slow down disk scanning as many non-existent files are searched for. If you change this value the family will be re-scanned automatically to detect any new members this may have made visible.

FAM_SIZE - Setting the file family size to use.
The file family size from LS-DYNA defaults to 7MBytes, but is sometimes set to some other value (using the X= parameter on the input line). D3PLOT can determine the member size of each family automatically by taking the larger of the first two members rounded up to the nearest Mbyte.

However you can override this value if, for some reason, the automatic method does not give the correct answer. Doing so causes the file family to be re-scanned automatically to detect any changes. Setting the value to zero effectively returns it to "automatic" mode.

SWAP_TITLE - Unscrambling endian-swapped titlesIf the analysis title appears to have every 4 letters reversed (ie ABCDEFGH = DCBAHGFE) then it has probably been (incorrectly) endian-swapped by your version of LS-DYNA. You can correct this by swapping between reversed and normal modes.

Setting these parameters externally

The parameters on this page may be set externally (or in the **OASYS** shell) with the following environment variables:

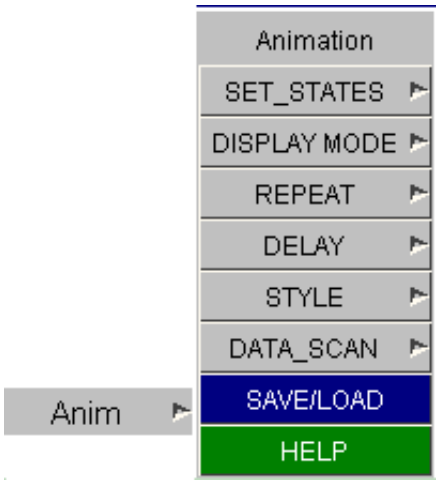
setenv FILE_SKIP 10 (Unix, C shell syntax)
FILE_SKIP=5, export FILE_SKIP (Unix, Bourne/Korn shell syntax)

```
setenv FAM_SIZE 7
FAM_SIZE=0; export FAM_SIZE
```

setenv SWAP_LSTC_TITLE true
SWAP_LSTC_TITLE=false; export SWAP_LSTC_TITLE

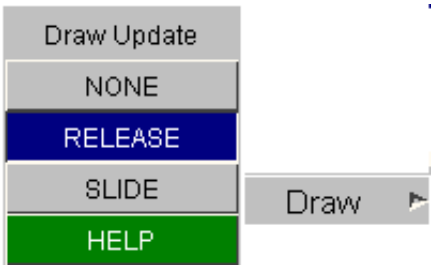
Windows users may set these variables in the **System**, **Environment** panel.

4.1.8 The Animation options popup menu



The **ANIM >** popup menu provides access to the options which control the extent, speed, and many other attributes of animation.
These, and many other aspects of animation, are described in Section 4.4.

4.1.9 The drawing vs. state selection menu.



The **DRAW >** popup menu controls how often (and if) the current state gets drawn when a new state is selected using the **STATE NUMBER** slider..

NONE	The newly selected current state is not redrawn at all.
RELEASE	The new state is drawn when you release the slider.
SLIDE	As you move the slider between states each one you pass gets drawn.

The default is **RELEASE** since this reduces drawing time for large models when selecting states.

Note that **SLIDE** can be used to scroll through (visually) an animation, but the same effect is achieved more easily using the frame slider bar on top of the graphics window: see Section 4.4.

4.1.10 The meaning of “fake” state #0

You may have noticed in some of the examples above that there is a state #0. This is a special state assembled within D3PLOT from the undeformed geometry with all displacements, stresses, etc set to zero. It is given the time 0.0. It exists for the following reasons:

- "Real" complete states in database files normally range from #1 to #n; but it is possible to create a database that only contains geometry, and has no complete states. Since all plotting commands within D3PLOT require a current state to be present a file with no states needs the "fake" state #0 be synthesised to permit plotting.
- State #1 may have the time 0.0, but may not contain the "undeformed" configuration: for example if an analysis started with a dynamic relaxation or from pre-defined displacements. In this situation reading in "fake" state #0 allows you to plot the undeformed configuration in all modes.

State #0 is ignored for purposes other than static plotting: it cannot be included in an animation, or used for time-history output

4.1.11 Support for analyses using “Adaptive Remeshing”

The adaptive remeshing facility in LS-DYNA, used primarily for metal-forming, generates output files in a sequence that is different to those from "normal" analyses.

Each remesh effectively constitutes a new analysis in which the quantity (and labelling) of nodes and elements in the remeshed parts will have changed. A new file family is generated at each remesh, and LS-DYNA flags these by appending "aa", "ab", etc to the output filenames for the 1st, 2nd, and so on families.

D3PLOT is able to detect this, so long as you give the name of the original analysis as the input file, and will automatically scan all the file families extracting their times, so that remeshed states are detected and made available. Thereafter you may use the programme in the usual fashion: selecting a new state automatically uses results from the correct file family, and capabilities such as animation work in the normal way.

There are a few minor limitations:

- Since each file family is a new analysis (it has different topology and geometry) D3PLOT has to maintain separate internal tables for each family. This requires extra memory, and you will find that overall memory consumption for adaptive analyses is greater than for a "normal" analysis of equivalent size.
  If you run out of memory trying to process all the remesh files together you may need to open its families individually. (See the MEMORY button for more information about viewing and modified database memory usage.)
- Blanking, colour, transparency, and related "property" changes are propagated through all families in an analysis sequence. However the situation can arise that an element in analysis A does not exist in analysis B. This is not an error, but it may lead to patchy looking plots. Therefore when changing visual properties of remeshed parts it is best to operate at the **PART** level (which will exist in all families), rather than on individual elements (which may not). Elements of parts that are not remeshed do not suffer from this restriction.
- Operations in **XY_PLOT** which imply collecting data across families will not work. It is not possible to extract "time-history" data in a consistent fashion across remesh (ie file family) boundaries.
- Using time interpolation for animation will only work within a remesh family: it is not possible to interpolate across remesh boundaries since the process requires nodal and element values to be interpolated between adjacent states. If you attempt to do this you will get a warning message, and the animation will "jump" to the first state in the new family.
- Reference geometry will not work in conjunction with adaptive remeshing, for the same reasons that interpolation will not: reference between two incompatible states is invalid

4.1.12 Support for file types other than “d3plot” transient analyses from LS-DYNA

D3PLOT is capable of reading the following file types and formats:

<name>.ptf **d3plot** **d3drlf**	Transient analysis data from Dyna3d, including coupled thermal/structural and implicit analyses. ("**d3drlf**" files are generated by dynamic relaxations.)
<name>.ctf **ctfile**	Contact force database from Dyna3d
<name>.xtf **xtfile**	"Extra" time-history file from Dyna3D. Only extra topological information is extracted from this file, no results are read from it.
<name>.htf	Thermal only output file (ex Topaz3D)
d3eigv	Eigenvalue (modal) analyses from Nike3D
n3plot	Transient analyses from Nike3d

D3PLOT can also read analyses from Nastran for a range of solution sequences (static, modal, forced response, etc) if they have been converted to "**.ptf**" files via a proprietary piece of software. At present this capability is under development - contact Oasys if you require more information.

Click here for the next section

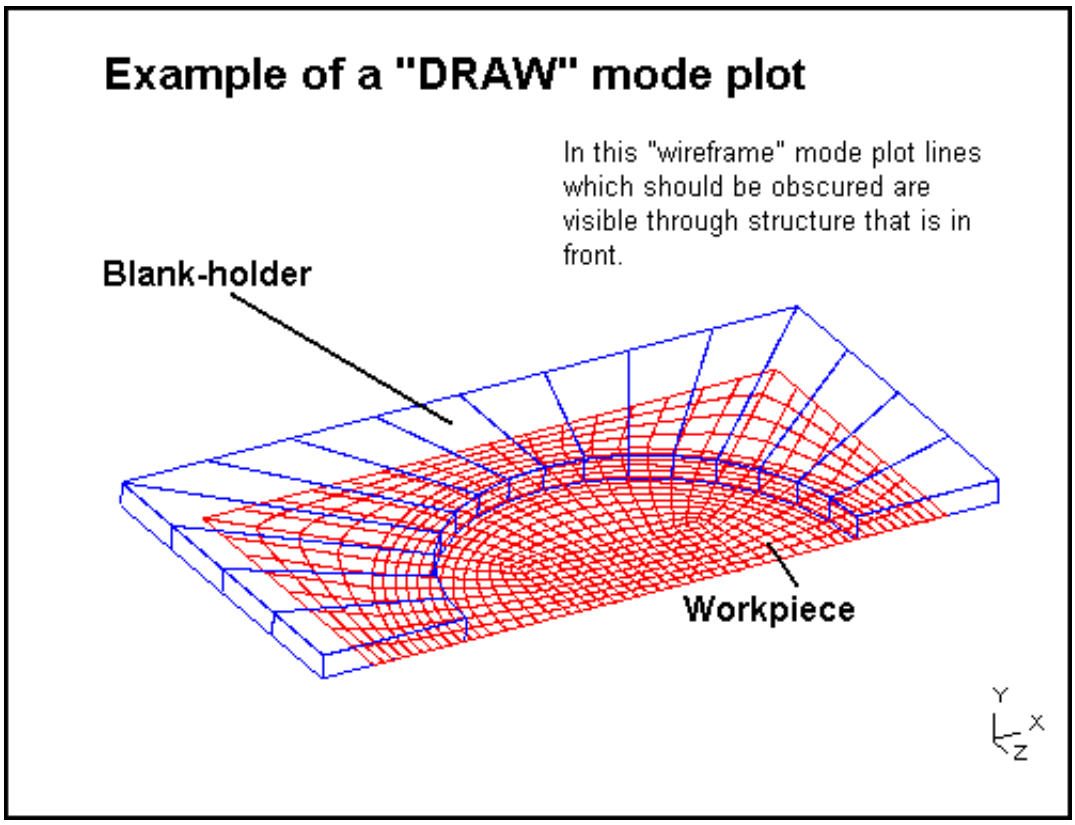
4.2 Displaying geometry and results.

4.2.1 Drawing commands that do not plot data

These commands are always available in all plotting "modes" since they do not display any data. They form the top row in this figure.

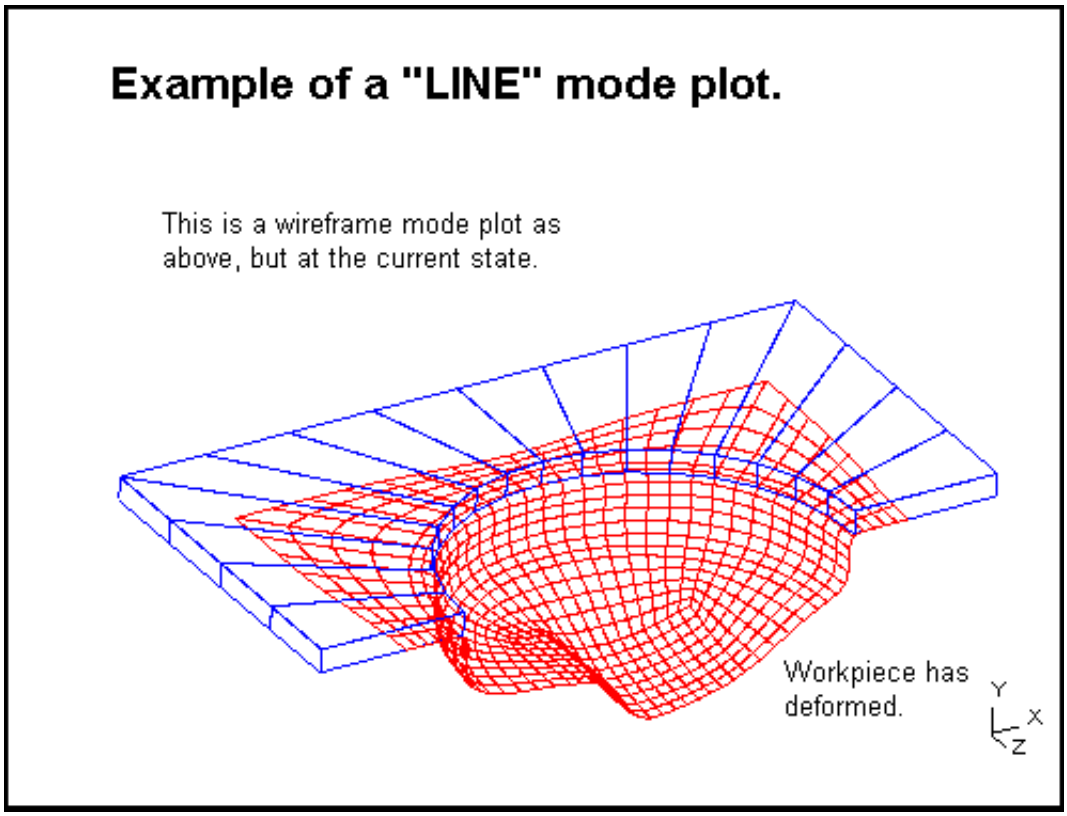
DR (alias **DRAW**) Draws the undeformed geometry in wire-frame mode, the current in-core state is irrelevant. No time is shown since none is associated with the undeformed geometry.

"Wire-frame" displays do not obscure hidden lines.


Example of a "DRAW" mode plot
In this "wireframe" mode plot lines which should be obscured are visible through structure that is in front.
Blank-holder
Workpiece
Y
X
Z

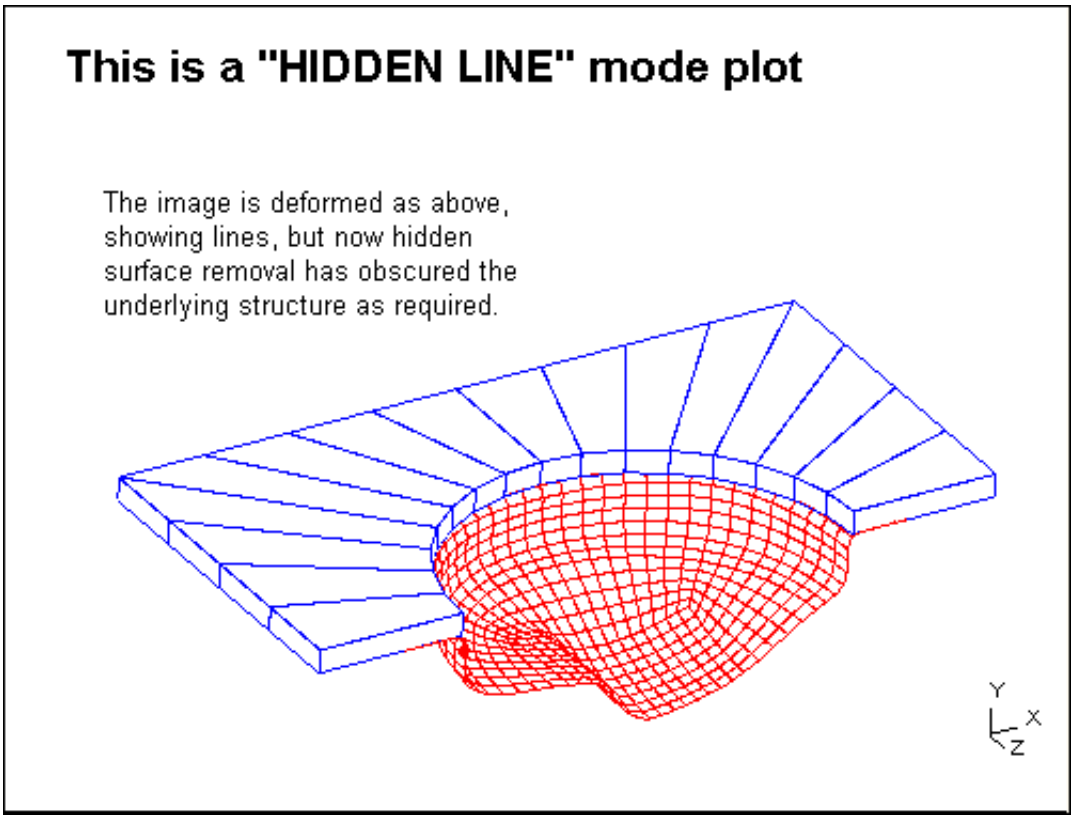
LI (alias **LINE**) Draws the current in-core state in wire-frame mode.
A time is now shown (in this case 4.6ms) since this represents data at that time.

Wire-frame mode still exposes lines which should be hidden.



HI (alias **HIDDEN_LINE**) Draws the current in-core state in hidden surface mode.

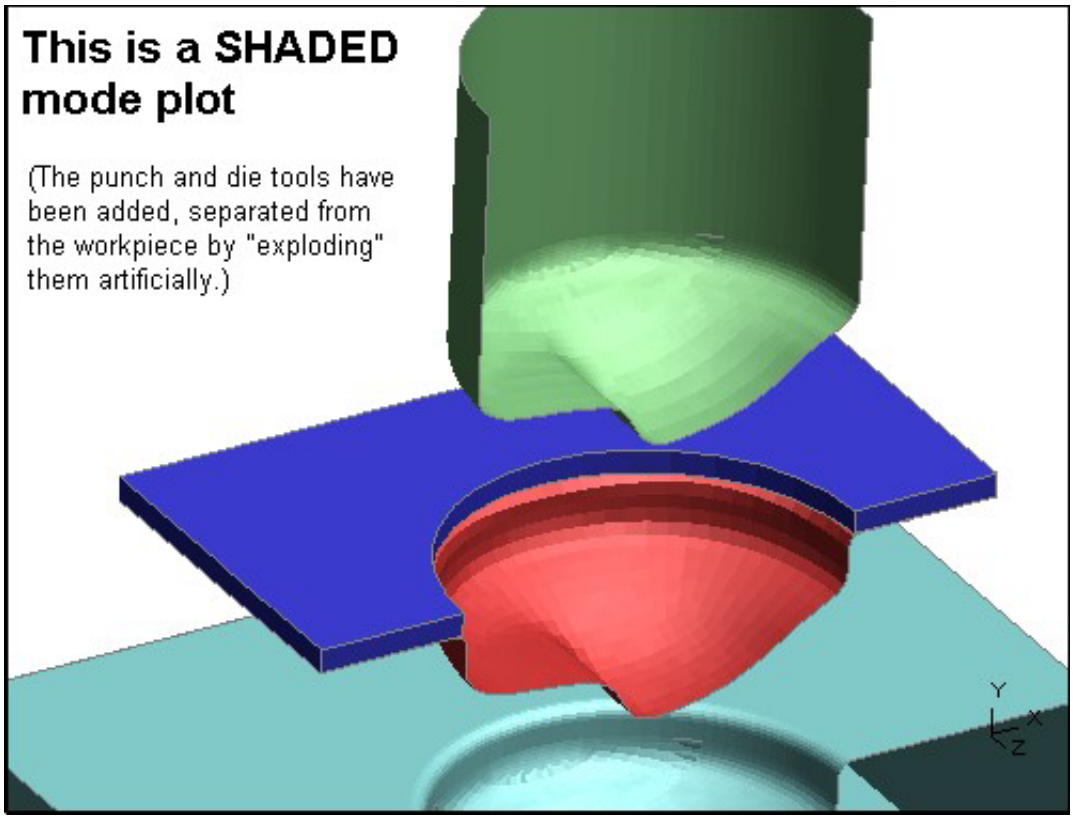
This plot shows the deformed shape at 4.6ms as before, but now hidden lines have been removed .


This is a "HIDDEN LINE" mode plot
The image is deformed as above, showing lines, but now hidden surface removal has obscured the underlying structure as required.
Y
X
Z

SH (alias **SHADED**)

Draws the current in-core state in lit and shaded "greyscale" mode, implicitly with hidden surfaces removed. The command-line equivalent command is **`[Greyscale] GO.`**

Shading has been applied, assuming a light source at the observer, and hidden-surface mode is implicit: any hidden lines will be removed.


This is a SHADED mode plot
(The punch and die tools have been added, separated from the workpiece by "exploding" them artificially.)
Y
X
Z

4.2.2 Visual Controls

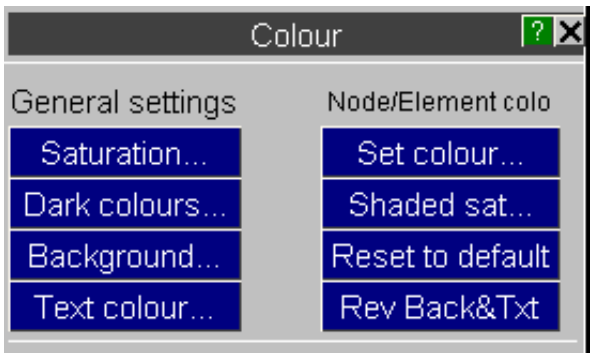
4.2.2.1 **COLOUR**: Controlling the colours used for the display

Note that changing element and part colours can also be achieved, usually more easily, using:

- The **Properties** panel, where all visual properties can be changed, saved and restored.
- "Quick Pick" screen selection, which is by far the quickest method.

The **COLOUR** command controls general colour usage in D3PLOT except the assignment of contour (see Section 4.3) and hidden-line overlay (see Section 4.2.4.6) colours.

Note: *In release 8.0 onwards many of its* **Node/Element** *colour functions are duplicated in the more user-friendly and capable* **PROPS** *box (see Section 4.2.2.3 below).*



"General settings" commands that affect all plotting modes are:

SATURATION... Controls the saturation of all colours in the range 0% (grey) to 100% (fully saturated). Normally all colours are fully saturated (ie 100%) to give the brightest possible display, but you may need to desaturate colours in some circumstances, for example when capturing frames for a video.

DARK_COLOURS... Is used to lighten colours for colour printers. Most colour printers use cyan, magenta, yellow and black inks and they tend to render darker colours such as blue and magenta too darkly.

This command lightens these colours preferentially so that colour plots look better, even though the display may look strange. Its default value is 0% (ie no lightening), and it may range to 100% (which will turn blues into white). You will need to experiment with your plotter to find the best value, a suggested starting point is 50%.

BACKGROUND... Sets the graphics window background colour. By default this is black, but you can choose from a range of standard colours, or make your own user-defined shade using the colour **PALETTE** (see below).

TEXT_COLOUR... Controls the colour used for text (ie clock, header, etc). By default this is white but, as with the background, you can make this a standard colour or a user-defined shade using the colour **PALETTE**.

Text and background colours may be chosen from one of the 16 standard colours shown here. Alternatively you can use:

Resets the relevant colour to its default. That is white for text, and black for the screen background.

To create any colour that your hardware can support using the colour mixing **PALETTE**.

Node/Element colours: Setting the colour of nodes, elements and other items.

A default colour is assigned to every entity type in D3PLOT as follows:

Standard 14 colour sequence:

1, 2 & 3D elements } Use the element part number or segment surface number to obtain colour from the standard sequence here on a modulo 14 basis.

Contact segments }

By default these colours are used only in plots that do not imply data plotting, that is:

DRAW & **LINE** (Wireframe), **HIDDEN** (Hidden wireframe) and **SHADED** (Solid shaded and lit)

Joints	Use their type number (#1 = spherical ... #7 = locking) to give a colour in the standard sequence above
Stonewalls	Use Magenta
Lumped-masses	Use Cyan
Nodes	Use White

In all other plotting modes the element "overlay" colour (controlled by the **OVERLAY** option) is used instead. You can change these colours as follows:

SET_COLOUR... Lets you define a new colour for a range of entities. To use it first define a colour from the options given, then select the range of entities to which these are to apply. Note that colours can also be changed using Quick-Pick (see section 3.5)

SHADED_SAT... This has the effect of desaturating "shaded" images, which gives them a more pleasant appearance. It has no effect on data plotting modes (eg **SI**, **CT**) or on wireframe/hidden element borders.

Note also that the "global" **SATURATION** control remains. It may be used to desaturate all images. The effects of the "global" and "shaded" saturation controls are additive.

RESET_TO_DEFAULT Resets all entity colours to their standard D3PLOT defaults.

REV BACK&TXT Reverses Text and Background colours.

4.2.2.2 **OVERLAY...** Controlling the hidden-line overlay of element borders on data plots.

This figure shows the hidden-line overlay control panel.

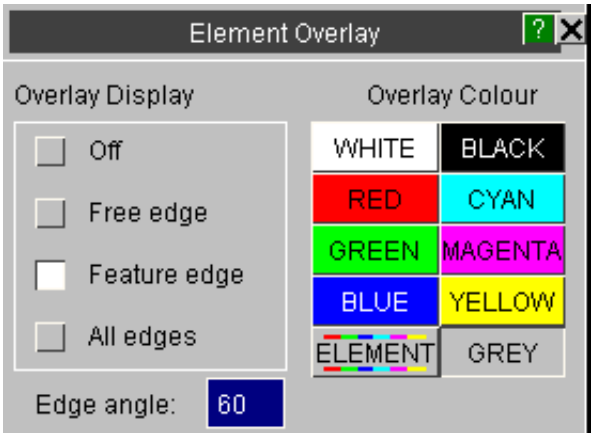
This panel controls whether or not a hidden-line overlay is superimposed on plots, and its attributes (when drawn).

Overlay Display Controls whether and how element overlay is drawn. It does not affect the current attributes.

Overlay Colour Obviously sets the colour to be used. The default is white, but other standard colours can be chosen. **ELEMENT** colour means use the **Overlay** colour of each element - see the **PROPS** box in Section 4.2.2.3.

Hidden-line overlays apply as follows:

- Overlays affect all data plotting display modes, and also **SH** shaded (**GREYSCALE**) plots.
- Their colour is normally fixed, but using the **ELEMENT** option, in conjunction with the **PROPS** panel, permits any permutation of overlay colours to be used.
- The display of edges may be one of:



OFF No element overlay is drawn

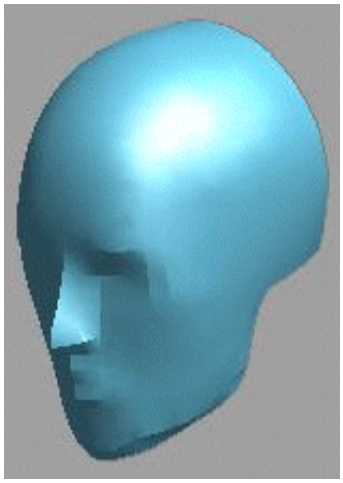
Free edge The topological free edges are drawn

Feature edge Free and "feature" edges are drawn

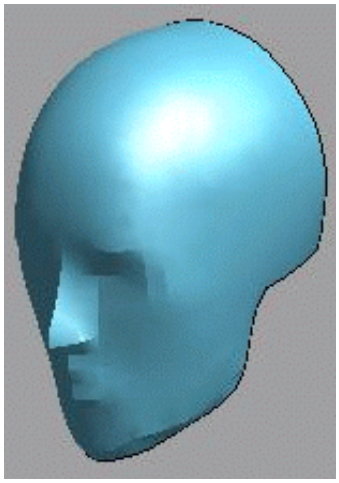
All edges All element borders are drawn

The **Edge Angle** is the angle between adjacent facets at which an "edge" is deemed to occur. It affects both Feature edges and also smooth shading. The default of 60 degrees is satisfactory in most cases, but to obtain more edges reduce this value. Values approaching 180 degrees will eliminate edges altogether.

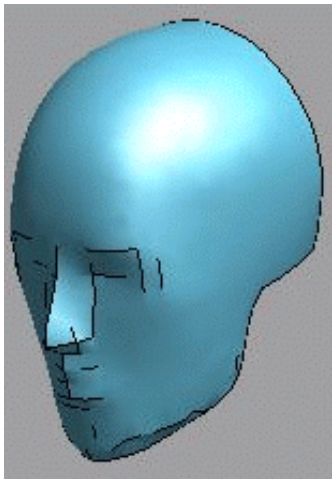
The effect of these various edge drawing options is shown in the four images below.



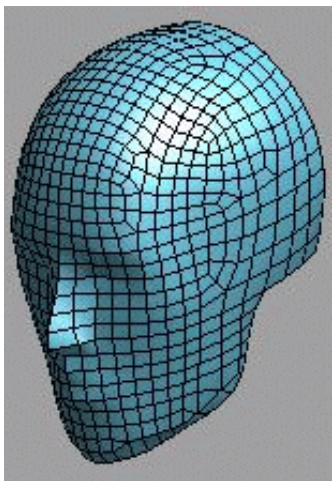
This image has no overlay at all



This image has only free edge overlay, which is drawn only at the element edges at the back of the head.



This image has feature line overlay with an "edge angle" of 20 degrees, showing detail around the mouth, nose and eyes.



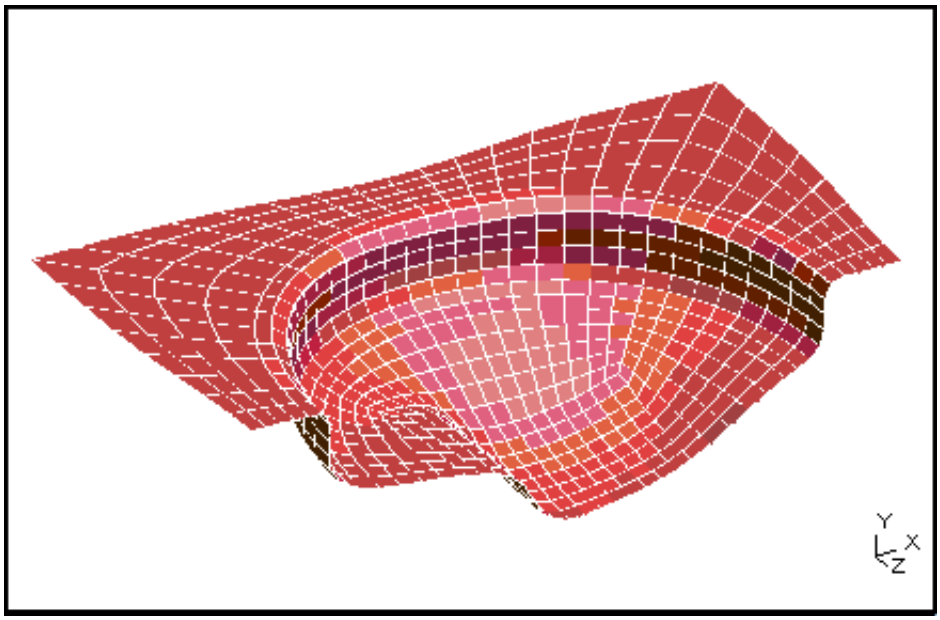
This image shows all element borders.

Controlling overlay quality in 3D mode.

Hidden-line removal in Z-buffered 3D graphics presents a problem to the programmer because the edges round elements will look "patchy" if they are drawn (correctly) co-planar with the element infill. Therefore overlay has to be raised slightly towards the observer (in the screen space Z dimension) to lift it above the surrounding elements, and the algorithm which calculates this "Z lift" dimension is usually satisfactory for all normal viewing parameters. However there are a few cases in which it can go wrong:

This image demonstrates the "patchy" overlay that can occur if the "Z lift" dimension is insufficient to raise it above the surrounding element infill. This does not normally happen, but it can occur if:

- Perspective is turned on
- And the perspective distance is very small



To fix this problem use <**right ctrl**> + <**mouse button**>. Moving the mouse up the screen will raise the overlay towards you, moving it down away. The mouse button used controls the speed of "Z lift" change:

<*Left mouse*>	Produces slow change
<*Middle mouse*>	Moves it more quickly
<*Right mouse*>	Makes large changes

If you subsequently revert to a more "normal" viewing distance you may need to reset the "Z lift" to get an acceptable image quality.

4.2.2.3 **Properties**: Controlling colour, drawing style, transparency, lighting attributes and overlay of entities.

It is important to understand the distinction between "Properties" and "Settings" in D3PLOT:

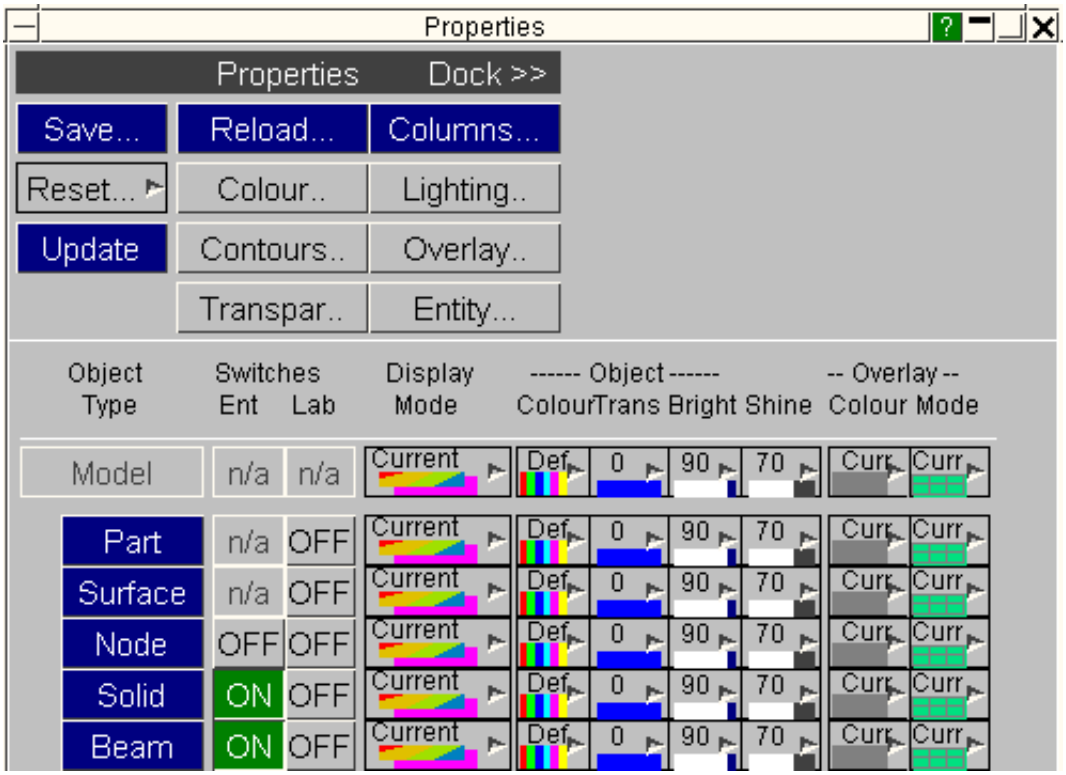
Properties	Are attributes of a model, for example part colours.
Settings	Are attributes of the programme and menu interface, for example data component.

A fuller description of these differences is given below.

The **PROPS** box duplicates the colour setting capabilities of the **COLOUR** command above (which is kept for backwards compatibility), but provides many more capabilities for improving the visual properties of plots:

- Explicit visibility (blanking and entity switch) control.
- Mixed display modes (contoured/shaded/hidden/wireframe).
- The ability to label items selectively.
- Colour setting.
- Transparency setting.
- Lighting attributes (diffuse brightness and shininess).
- Overlay colour and style (solid/free edge/omitted).

These properties may also be changed using Quick-Pick.All of these capabilities are available at <model>, <category (eg part)> and <individual item> level; which makes it possible to tune plots for presentation to any degree. Model properties can be saved and restored, and even applied to different models: see below. Due to the width of the menu, the option is provided to **UNDOCK** the menu. This moves it from being docked in the menu area to floating in the Grahics area. **DOCK** will reverse this process.



Selecting the level at which to operate: **<Model> <Category> <Entity>**

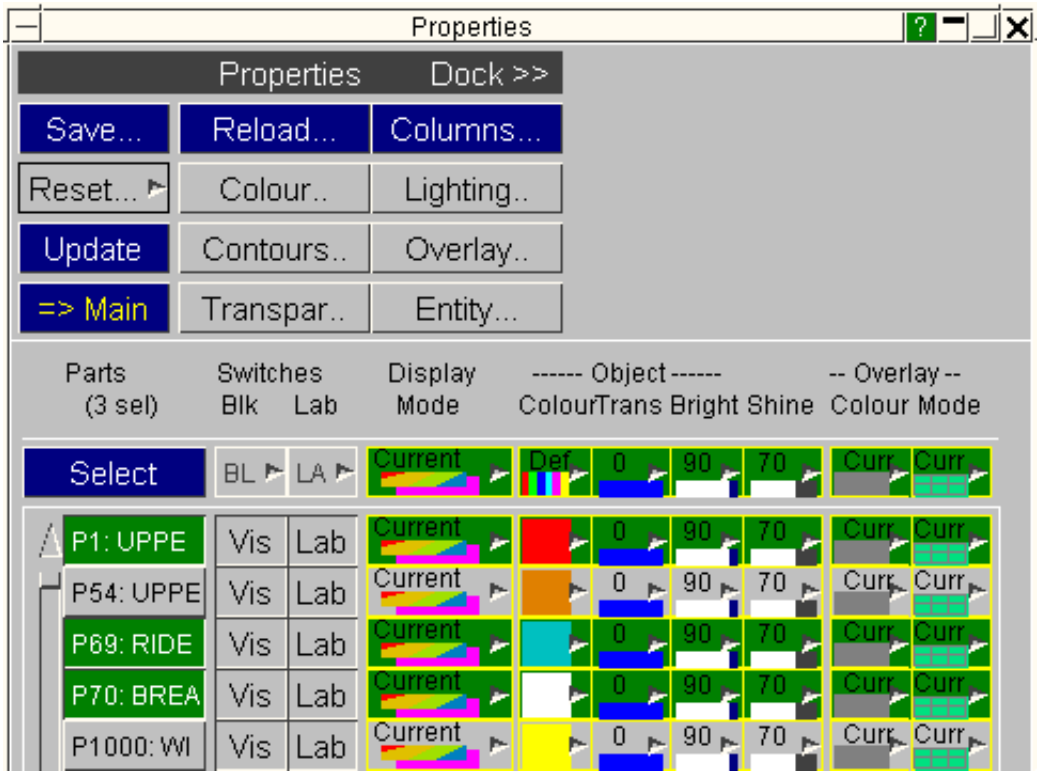
The **PROPS** box operates at two levels: the top one applies to the whole model, and all its constituent categories; the lower one applies to individual items in a category. In this example the user has chosen to operate on individual parts.

Selecting **PART** at the top level...

...results in the **PART**s sub-panel being displayed.

It is also possible to **SELECT** any permutations of items, using the standard D3PLOT selection process or by clicking on their id button. Selected items have their background changed to green (here parts 1, 3 and 4 have been selected).

Operations applied to the top row buttons are then applied to all selected items .



Columns.. Controls which attributes columns are shown

Because the **Properties** box can be configured to contain a lot of information, which can look confusing, the

Columns... menu can be used to control which columns of attributes are shown. There are four of these, which may be turned on/off independently:

Entity/Label Switches

At Top level the **Entity** switch controls whether or not that category is drawn at all. (See Section 6.6) The **Label** switch controls whether or not it is labelled at all. (See Section 6.5) At sub-panel level the "Entity" column is replaced by **BLANK**: each element may be (un) blanked individually. (See Section 6.1) Similarly the **LABEL** switch may be applied individually to each element.

Display Modes

At top level each category, or the whole model, may be set via its popup menu to one of WIREFRAME, HIDDEN, SHADED (See 4.2.1 for what these mean) or Current (Elements are drawn in the currently selected display mode, whatever that may be). At sub-panel level each entity can be set separately in a similar way. It is possible to select any mode for any element, and to display them in combination. For example you could have a mixture of shaded, wireframe and contoured display in different areas of a mesh.

Solid-filled (ie **CT** or **SI**) contours are not shown on elements rendered in wire, hidden or shaded modes. However vector data (**LC**, **VEL**, **VEC**, **Criterion**) is superimposed on the current display mode of elements. For example velocity vectors may be drawn on shaded elements.

Object Attributes

As before changes may be applied to whole categories at the top level, and to individual or selected items at the sub-panel level. There are four "attributes":

Colour: Select an explicit colour, or return to default.

Transparency: Transparency only applies to 2D and 3D objects, and by default they are all totally opaque. However you may set any such entity's transparency on the range 0% (opaque) to 100% (fully see-through) in increments of 10%.

Brightness: Setting diffuse brightness

Shininess: Setting specular brightness

Lighting is discussed in more detail in Section 4.2.3 below.

Overlay Attributes

Overlay Colour: Select an explicit overlay colour, or default.

Each entity maintains a separate overlay colour, distinct from its "current" colour. The **Default** overlay colour is the same as the native element colour, but the two are stored separately and may be quite changed independently.

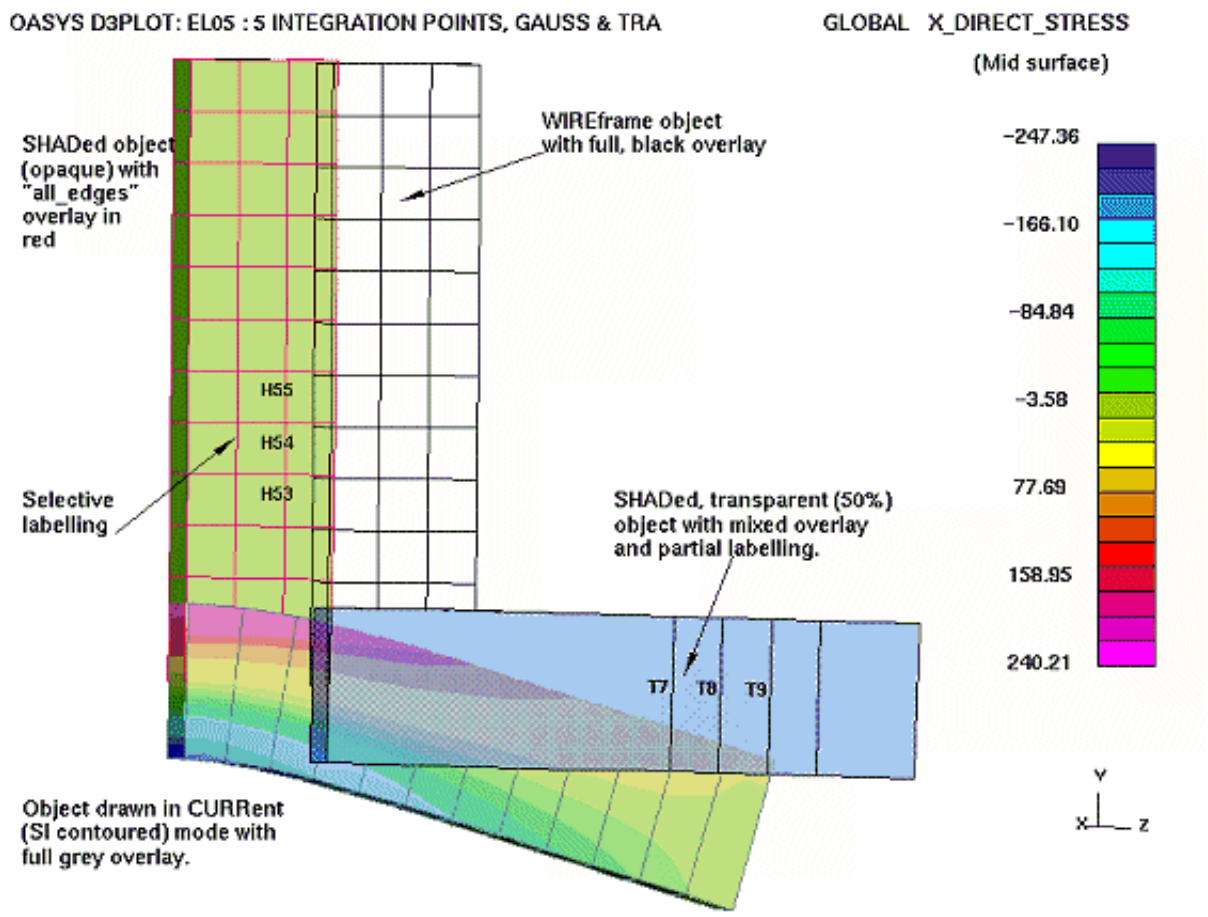
Overlay Mode:

Overlays may be drawn in one of three styles:

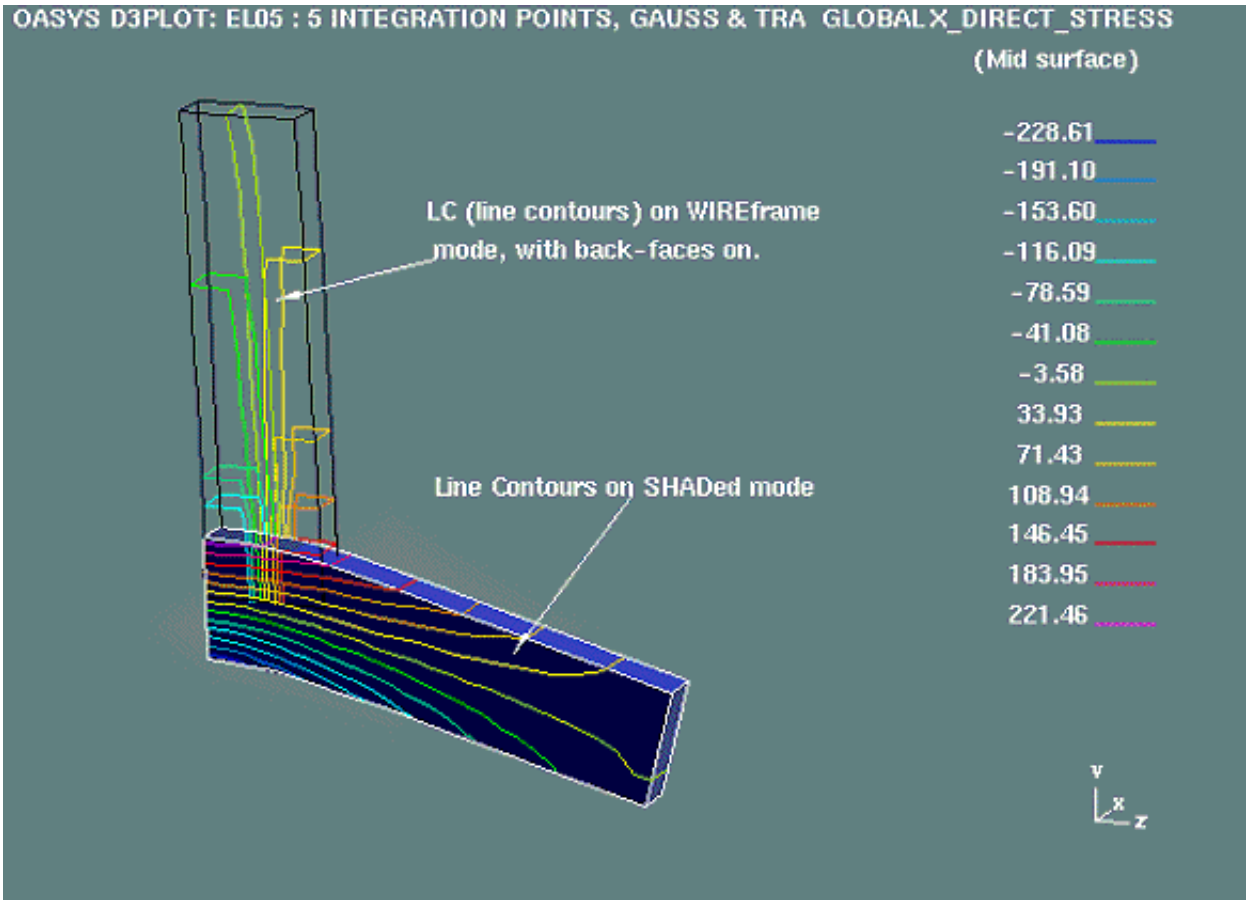
NO_OVERLAY: Not drawn at all.
FREE_EDGE: Only free edges are drawn.
ALL_EDGES: All edges are drawn.

When drawing the more restrictive of the main **DISPLAY_OPTIONS**, **FREE_EDGE** switch and the "local" element ones is applied: if either eliminates edges they will not be drawn. See section 4.2.2.2 for more detail.

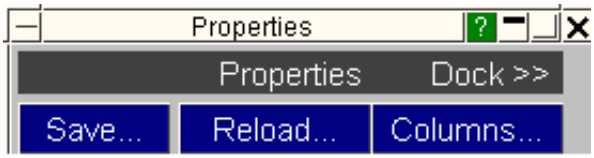
So what does all this mean? Here is an example which combines transparency, different modes of plotting, selective labelling and various overlays to show what can be achieved.



Here is another example which shows how "vector" mode plots (here **LC**, but also **VEL**, **VEC** & **Criterion**) can be superimposed on shaded and wire-frame rendering.



OPTIONS... SAVE and **RELOAD**: Saving and restoring properties.



Setting up properties (colour, transparency, etc) for a model can be time-consuming, and it is useful to be able to preserve them for subsequent runs.

- **SAVE** writes the complete property list for the whole model into a "properties" (`.prp`) file.
- **RESTORE** can read this in at a later date and restore all the saved settings.

You can create any number of property files, each will save the current status, and these may be read in at any time to update the current display.

Property files can be applied to different (but similar) models:

- All information in a property file is written using ***external*** item labels.
- When read back in these labels are mapped onto the items in the current model that have these labels.
- If an external label is encountered that has no internal counterpart it is ignored.
- Similarly those items in the current model which don't appear in the properties file are left unchanged.

Therefore you can apply a properties file to any model and, provided that it is not too dissimilar to the model from which the file was written, the effect should be to restore your settings almost completely.

Reading and writing properties files elsewhere in D3PLOT

Properties files can also be written and read back in from the **UTILITIES, SETTINGS_FILE** panel. This is described in section 6.9.10.

They can also be read in at the same time as the model is input if the **Read PRP file** box is checked on the file input panel - see section 4.1.1

What is the difference between a Properties file and a Settings file?

A ***properties*** file contains information about the model properties: colour, transparency, labelling, visibility, display modes, etc. It is window independent.

A ***settings*** file contains information about the D3PLOT window settings: background colour, data component, contour levels, etc. It is model independent.

Properties and Settings files are forwards, but not backwards, compatible.

Both of these file types evolve with successive releases of D3PLOT. A newer format file will not read into an older version of the code, however an older format file may be read into a newer version of the programme and - usually - will function normally.

There are some minor exceptions: some overlay attributes in a properties file from release 9.0 of D3PLOT may not translate properly to a newer release. This is not so much because of file incompatibility, but rather because of changes to the inner workings of the code itself which make the "old" attributes invalid.

If you have problems with incompatibility please contact Oasys for help and advice.

Click here for the next section

4.2.3 The **LIGHT** panel.

The lighting model in D3PLOT is a simplified version of the diffuse and specular models found in most graphics hardware(1), and in standard computer graphics texts(2). It is a compromise between speed and appearance that is adequate for rendering engineering plots and simple presentational material, but its results fall short of those from a ray-tracing package: shadows and reflections are not provided, and surface properties are crude.

Geometry can be exported to external rendering packages via the **UTILITIES**, **VISUALISATION** menu for those cases where higher quality images are required.

The lighting model is best thought of as having three parts:
1. The light sources.
2. The way in which the objects are shaded.
3. The "material" properties of the lit objects.

The Light panel ontrols the following lighting and shading attributes:

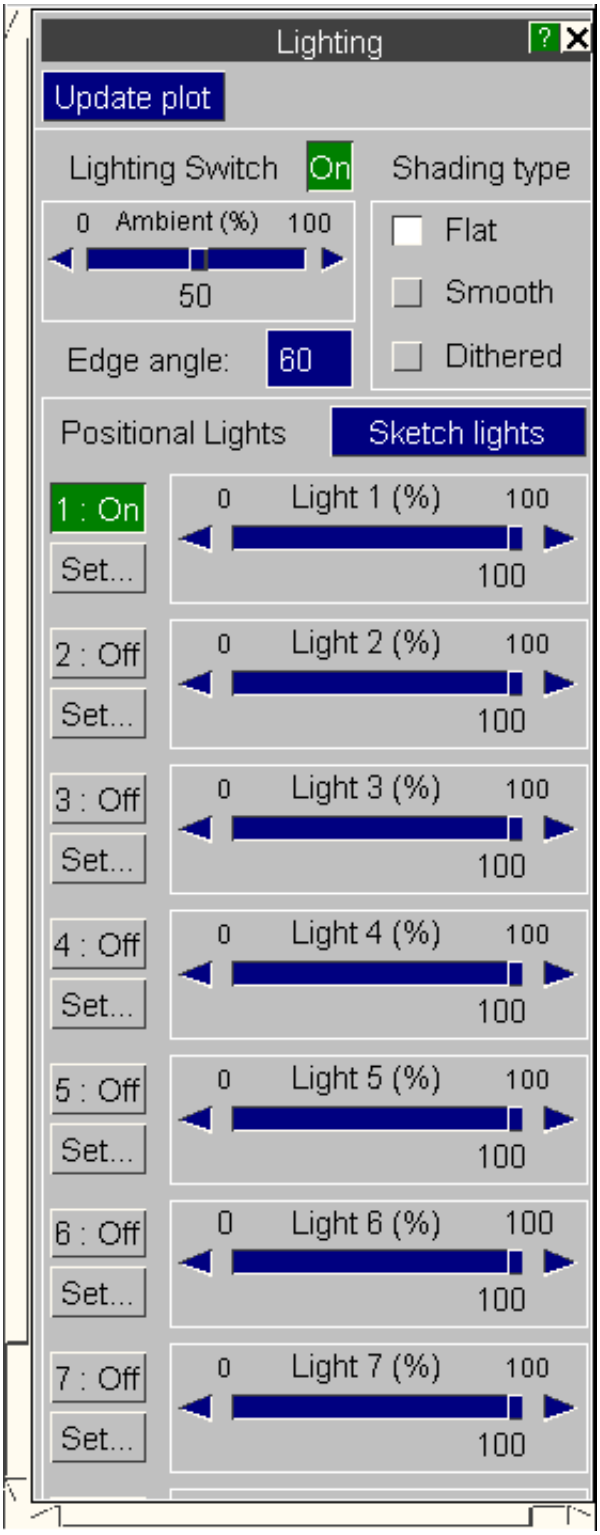
Lighting switch: Global on/off for lighting.

Point light sources: On/off switch, position, intensity, space system (model/screen).

Ambient light: Intensity.

Shading type: Flat / Smooth / Dithered.

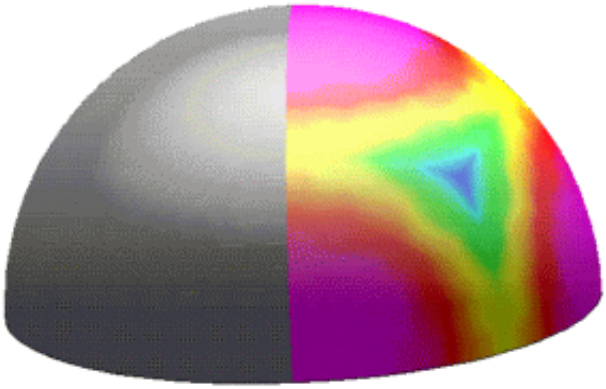
Edge angle: Angular difference limit for smooth shading across adjacent facets.



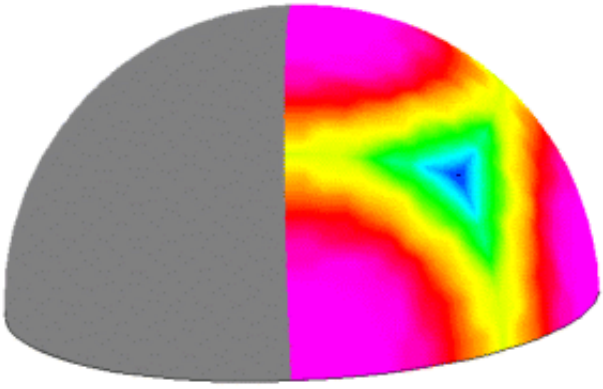
Lighting Switch
Global lighting on/off switch

By default lighting is ***ON***, and a plot implying lighting will use the current attributes. If lighting is turned ***OFF*** all formerly lit elements will be drawn at the full colour intensity with no shading effects.

Turning the switch on/off does not change stored lighting or shading attributes in any way.



Lighting Switch ON

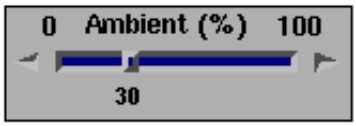


Lighting Switch OFF

These two examples show how a hemisphere, half **SI** contoured and half **SH** shaded, responds to the lighting switch. Note how turning lighting off destroys any perception of shape or depth.

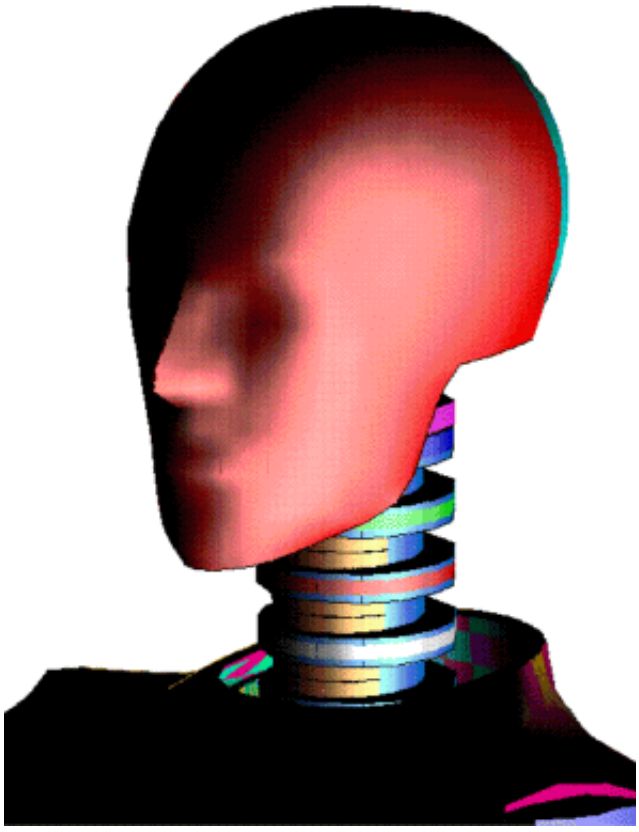
(The effect on the right could also be achieved with lighting ***ON*** by setting the **Ambient** light to 100% and having no point light sources active.)

Ambient light
The %age of "black body" light.



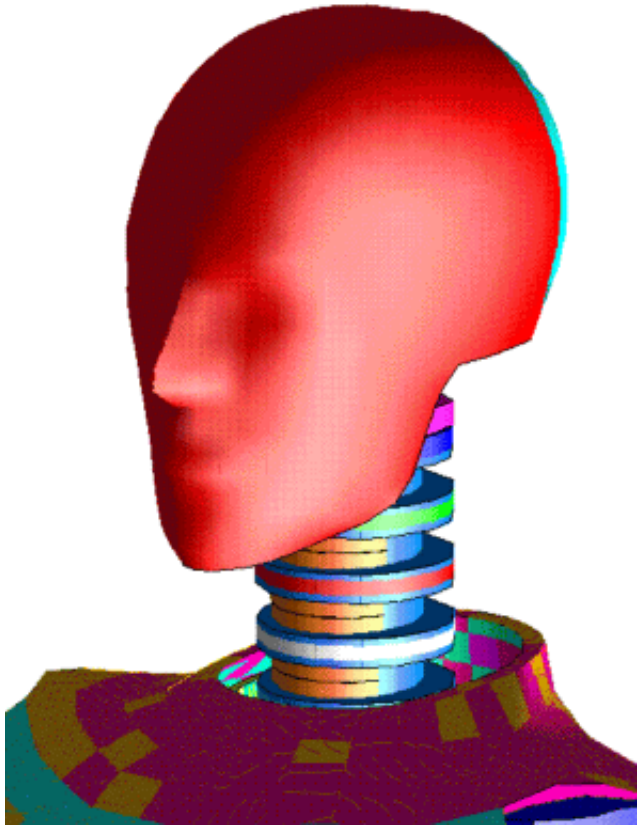
If only point light sources are used this simple lighting model produces images reminiscient of pictures from space: strongly directional lighting with no detail in shadow areas. Therefore the concept of **Ambient** light, analagous to "black-body" radiation coming from the universe at large, is added to fill on lowlight areas.

This is not realistic in itself, but in practice most scenes have their shadow and low-light areas filled in by reflected light from walls, floors, etc; and it provides a good approximation to this.



This image has a single directional light to the right, and has the **Ambient** light set to 0%.

Note how the lowlight areas are extremely dark and contain no detail.



Here is the same image with the **Ambient** light set to 40%.

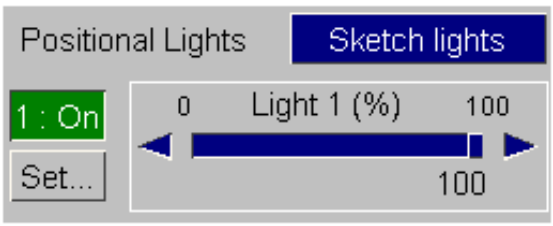
This has filled in the lowlight areas to some degree, but a higher value still is needed to illuminate some areas.

Why bother with ambient light? Why not just use more (and more realistic) point light sources?

The short answer is speed and simplicity. Ambient light is cheap to compute and easy to define, whereas adding point lights slows down image redraw speed. And, as any photographer will testify, getting the position and intensity of multiple light sources correct is not as simple as it seems. However you have 8 light sources to experiment with: feel free! (But note that some graphics hardware may not operate correctly, or may run slowly, with more than two light sources.)

The default in D3PLOT is a single directional light above and to the right of the observer's position, and an Ambient light level of 40%.

Point light sources (positional & directional)



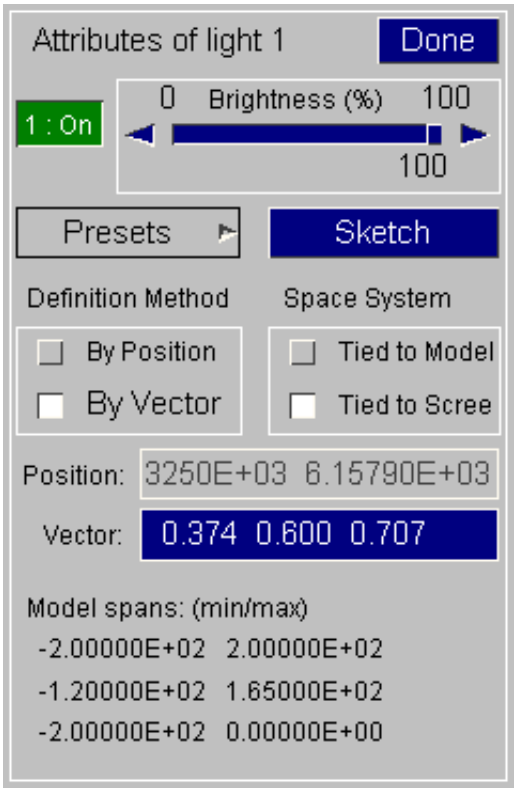
Up to 8 point light sources may be defined, each of which is independently controllable. At the top level of the lighting panel they may be turned on/off and have their intensity set from 0 - 100%. To change any other attributes use the **SET...** button.

SET... Setting detailed light attributes

The following detailed attributes for each light may be set:

OFF/ON Turns this light on or off.

Brightness Sets the light brightness in the range 0 - 100%. (nb: it is more efficient to turn a light **OFF** than to use 0% **Brightness**.)

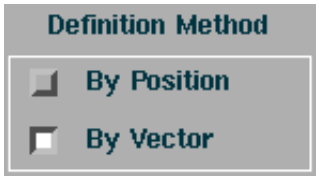


PRESETS >
Provides some pre-computed positions for lights.

These may not give exactly the locations you want, but they can form a good starting point.

(The default light in D3PLOT, light 1, is positioned at "Right Shoulder".)

Definition Method Positional or Directional (by vector)



Example of Positional light.

A "positional" light is defined by its location in space, and this method is generally used for lights close to the object.
This example shows a dummy with a positional light in its lap. (The light itself has been added artificially here, it would not normally be visible on a plot.)

Because the light is "local" it (correctly) does not illuminate the tops of the arms or the front of the legs.
Positional lighting is slightly more expensive than directional to compute, but is necessary if the true effects of lights close to the object are to be simulated.

Example of Directional light.

A "directional" light is defined only by its vector, and assumed to be infinitely distant (like the sun).
This is the same model as above, but now the light is directional, defined by a vector pointing through the previous light source position towards the dummy. (The vector arrow has been added artificially, it would not appear on a plot.)

The differences are clear: the whole of the front of the dummy has been illuminated, as have the surfaces which previously were dark.
This is a little bit cheaper to compute than the positional equivalent, as "local" vectors from facet to light source need not be computed.

Defined lights as Tied to Model and Screen space systems

Light Tied to Model space

<== **Before** ..transform .. **After** ==>

When a light is tied to **Model** space it is transformed along with the object.

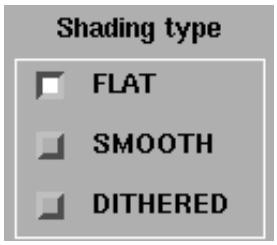
Light Tied to Screen space

<== **Before** ..transform .. **After** ==>

When a light is tied to **Screen** space it remains fixed in space as the object is transformed.

(In these examples the light sources have been added artificially to illustrate their positions. They would not appear on an actual plot.)

Shading type: **Flat**, **Smooth** and **Dithered**.



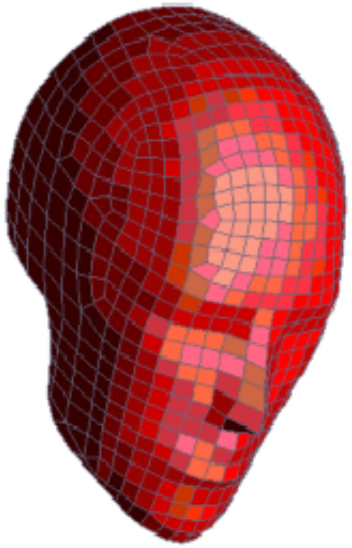
This controls how facets are shaded, which in turn affects the appearance of curved surfaces.

FLAT shading

The outward normal of each element face is calculated and used to determine a single lighting value. This is applied to the whole face giving the faceted appearance shown here.

This is quick to compute and, with a fine enough mesh, gives acceptable image quality.

(The mesh overlay has been added here to emphasise that each facet has a single flat shade.)



SMOOTH shading

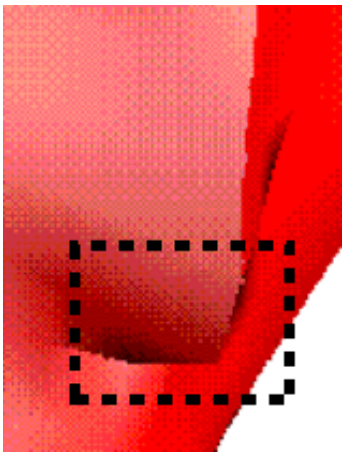
The outward normals at each vertex are averaged, making it possible to vary lighting smoothly across a surface. This technique is known as "Gouraud shading", and is only available in 3D graphics mode.

DITHERED shading (*2D graphics mode only*)

Because gouraud shading is not provided by 2D graphics drivers (or laser plotters) the technique of "dithering" has to be used to produce genuinely smooth shading under 2D graphics. This is done by drawing adjacent pixels in different colours to achieve an intermediate shade - trading of spatial against colour resolution.

Dithered shading is ignored in 3D mode, and should only be used in 2D mode if you are prepared to accept the longer computation and display times involved.

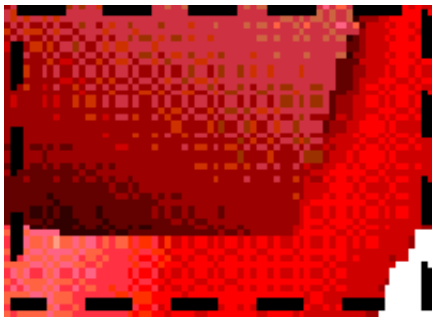
What dithering actually does, and how to fix problems that may arise from its use: (***2D mode only)***



On the left is an enlarged area of the dithered equivalent of the image above showing the bottom tip of the nose.

The dither pattern is just visible in the image on the left, and if magnified further (right) it becomes obvious.

There are only five different colours used in this image, yet it has been possible to show a wide range of shades. However spatial resolution has been lost in favour of colour range.



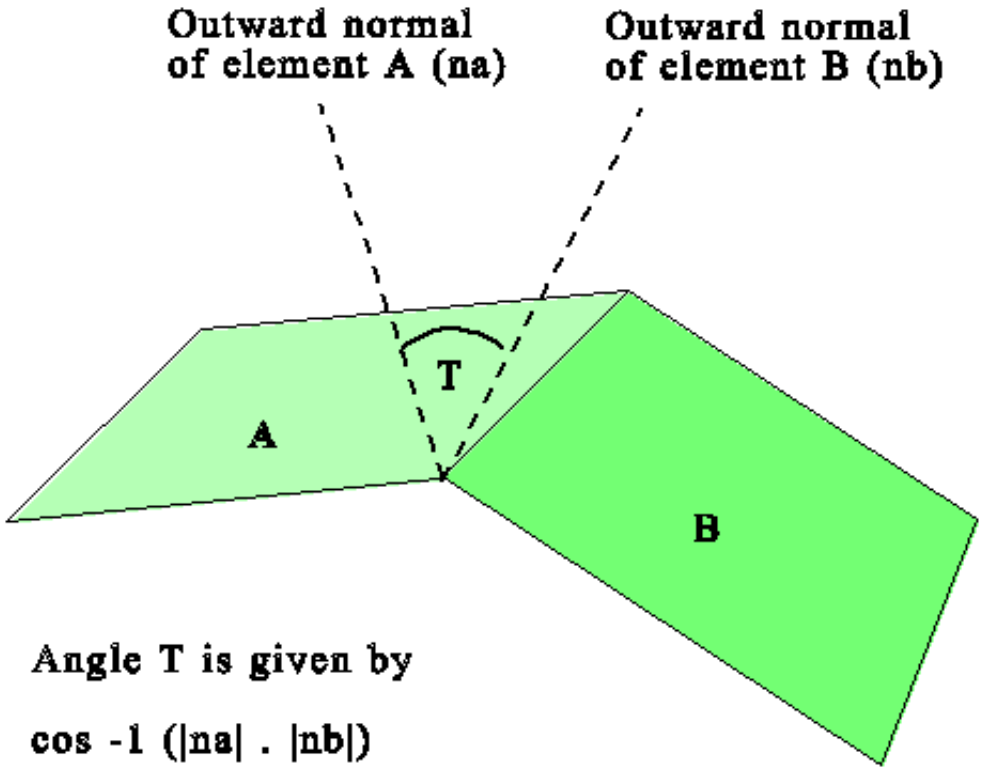
Dithering can occasionally cause problems when images are captured from the screen or laser-plotted. In particular there can occasionally be a "heterodyning" (beating) between the spatial resolution used for dithering on the screen and that used by the subsequent display device. This can show up as an apparent chessboard of large lighter and darker squares on the image, or as light/dark bands. Some software packages for manipulating bitmaps may show similar effects.

If this happens you may be able to fix it with one of the following:

- Try generating the image with a different screen window size, and hence a different scale. This may be enough to stop the "beating" effect.
- If you have been working at 8 bit-plane resolution, but your screen supports 24 bit-plane "true colour", then use that instead. At 24 bit-plane depth all possible colours that the human eye can resolve (2^{24} = 16,777,216) are available, and dithering is not required. The image will look better too.

Edge angle:

The angle at which smooth edges become sharp.



When smooth shading it makes sense to preserve some "sharp" angles, since most real-life objects have sharp as well as smooth edges.

This is done by computing the angle between the outward normals of adjacent facets at vertices, and only averaging if this is less than the current **Edge angle**. This effect is evident in the images of the head above: the ridge of the nose is "sharp" whereas the rest of the face is "smooth".

Edge angle can lie in the range 0 - 180 deg, (values > 90 deg are significant for 3D elements). The default of 60 degrees looks reasonably natural for most objects. 0 degrees is equivalent to Flat shading, and 180 degrees will eliminate all sharp edges.

The light sources themselves.

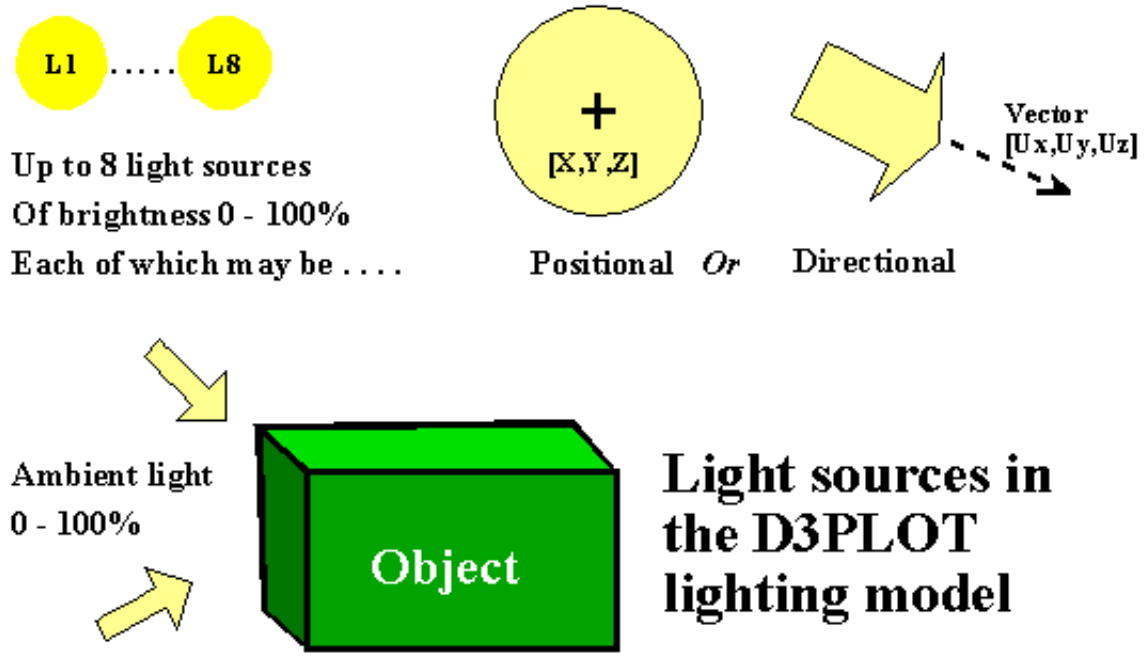
Up to 8 point light sources may be defined, each of which is independently controllable. At the top level of the lighting panel they may be turned on/off and have their intensity set from 0 - 100%. The OpenGL standard requires all implementations to handle 8 light sources, although it is often the case that only one or two cab be handled in hardware: don't be surprised if setting up many lights causes a sudden and dramatic reduction in graphics speed!

To change any other attributes use the **SET...** button. There are various preset options, and you can also specify your own light source positions or vectors.

Each may be "positional" (located at a specific point in space), or "directional" (infinitely distant along a vector). "Positional" light sources are more expensive for the hardware to calculate, and will slow down drawing, so only use them if you need to simulate the effects of a light close to your model.

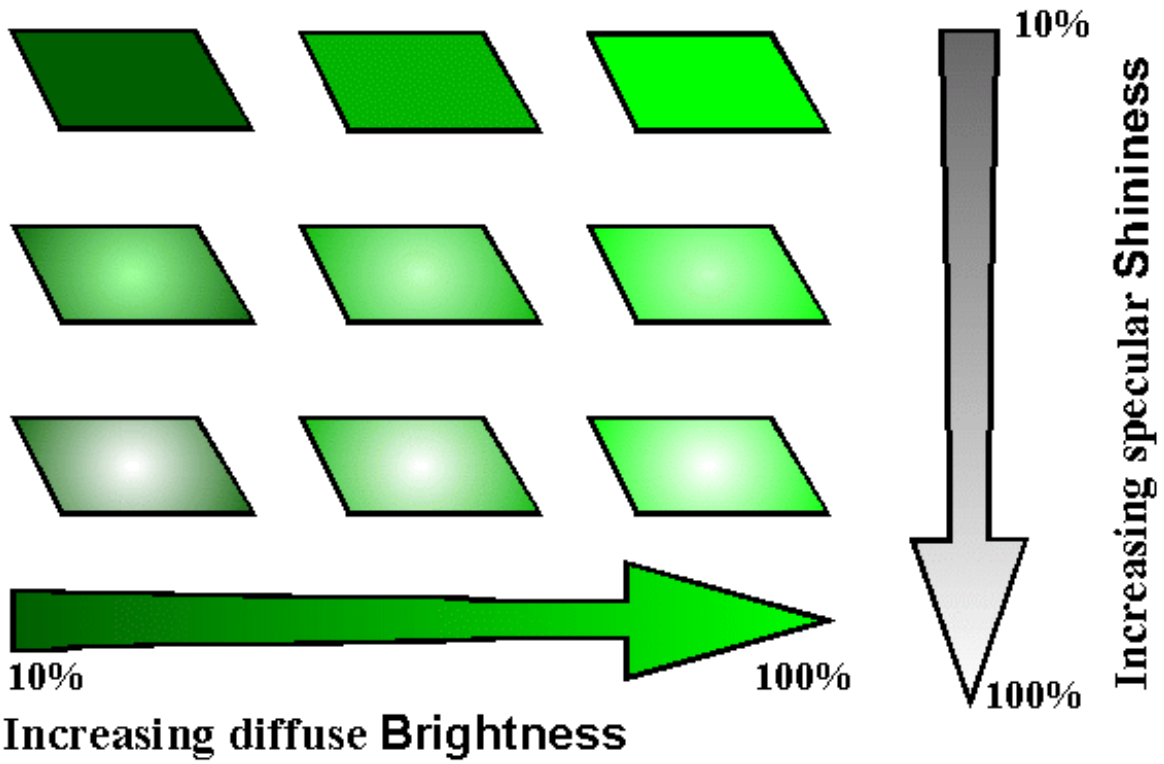
Light sources can be fixed in screen space (stay fixed when the model moves), or attached to model space (move with the model). There is no significant difference in graphics speed between the two options.

All lights are white.



The “material” properties of the lit objects

Brightness and **Shininess**



Brightness Controls how light or dark the colour of the object is when illuminated, but the effect is to add matt colour (not whiteness, which would make it look shiny).

Shininess Adds white highlights, but no colour, to make the object look shiny.

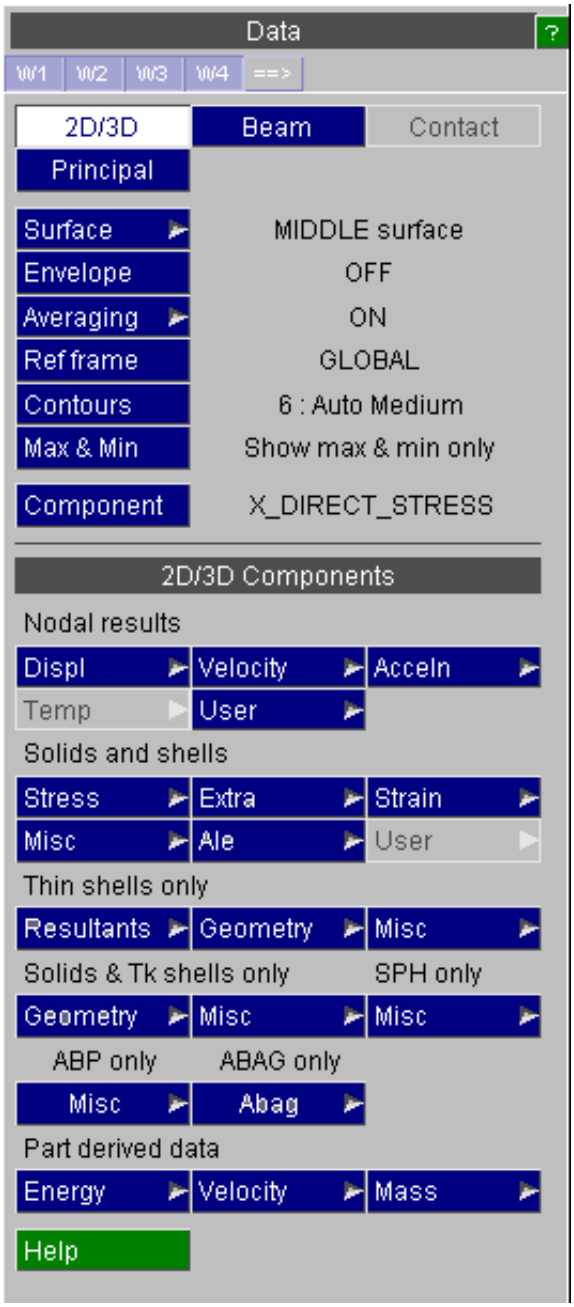
These attributes (of the object, not the lights) are set in the **PROPS** box - see Section 4.2.2.3.

1. OpenGL Programming Guide. Neider, Davis, Woo (ISBN 9 780201 632743)

2. Computer Graphics Principles and Practice. Foley, van Dam, Feiner, Hughes (ISBN 0 201 12110 7)

Click here for the next section

4.2.4 **Solids & Shells**: Plotting results on solids and shells (2D and 3D elements)



The figure (right) shows the Data panel in **Solids and Shells** plotting mode (in this case displaying the Components sub-menu).

This is the default data plotting mode in D3PLOT. It is available even if your model has no 2D or 3D elements since it contains the velocity vector plotting option (which is effectively data at nodes), although in this case some plotting options will be greyed out.

From release 9.3 onwards this mode also displays data for Smooth Particle Hydrodyanics (SPH) and Airbag Particle (ABP) elements.

4.2.4.1 **COMPONENT...** Choosing a data component to plot

This figure shows all the generic data components available for solid and shell plotting. If some are unavailable in your data-base they will be greyed out (for example the **EXTRA** components here).

Select the generic category, and then the specific component.

(For more information on selecting data components, and a more detailed description of what they mean, see Section 12).

4.2.4.2 **SURFACE...** Choosing a thin shell “surface” (intg pnt)

This figure shows the shell integration point selection box, which applies to stress and strain tensor derived results.

By default shells write results at 3 "surfaces":

Top	Is the outermost (most +ve local Z) integration point.	Note that when using the default Gaussian integration rules in LS-DYNA the inner and outer integration points of shells are ***NOT*** the "outer fibres" of the element. For a fuller explanation see the WARNING below.
Middle	Is the neutral axis, ie mid-plane.	
Bottom	Is the innermost (most -ve local Z) integration point.	

In this example shell output has been defined with <**maxint**> = 8, giving the option of each of the eight integration points in the element. In the conventional case, <**maxint**> = 3, the **"Layer"** column will not be shown. (See note 2 below.

Normally you will be interested in results at a given integration point, but it is also possible to extract the following values scanned from all integration points through the thickness of the element:

MAX_ALL	Finds the maximum (most +ve) value
MIN_ALL	Finds the minimum (most -ve) value
MAG_ALL	Finds the +ve or -ve value with the greatest magnitude. Result may be +ve or -ve since the calculation is: `if ( |val| > |curr| ) curr = val`

Shell integration point data written from LS-DYNA

There are two issues to be considered here:
1. The number of through thickness integration points in the shell element formulation.
2. How many integration points worth of data are written to the database files.

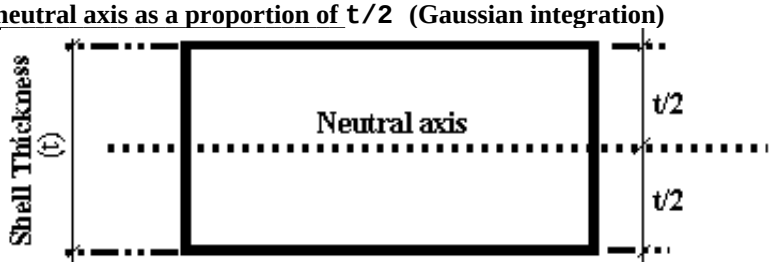
Unfortunately these two parameters are not directly related in LS-DYNA, and have to be set independently:
- Shell element through-thickness integration points are controlled on the *SECTION_SHELL card. (Composites are different)
- The number of "surfaces" output is controlled by parameter <**maxint**> on the *DATABASE_EXTENT_BINARY card

By default (<**maxint**> = 3) data at three "surfaces" (top, middle and bottom) are written for all shell elements, regardless of the actual number of integration points in any element formulation. Normally this is satisfactory, since in most models with significant plastic strain the detailed distribution of stress and strain through element thickness is not that important, but where models remain mainly elastic or where composites are used it may be necessary to set a different value. See Section 11.8.2.2 for a fuller discussion of these parameters and how they affect output.

WARNING: In LS-DYNA analyses the top and bottom shell "surfaces" are **not** the outer fibres of the element if the default Gaussian integration scheme is used: they are located some way in from the outer fibres.

The following table shows the location of the outermost integration points, as a function of shell half thickness **(t/2)**, for the most commonly used numbers of points.

No of Points	Distance from neutral axis as a proportion of t/2 (Gaussian integration)
1	0.0 (membrane)
2	0.577
3	0.775
4	0.861
5	0.906
6	0.932



Where you have written an odd number of integration points to the output file the "mid surface" will be the mid point.

Where you have written an even number of points it will be the average if the two "middle" values. For example if you write 6 points it will be averaged from #3 and #4.

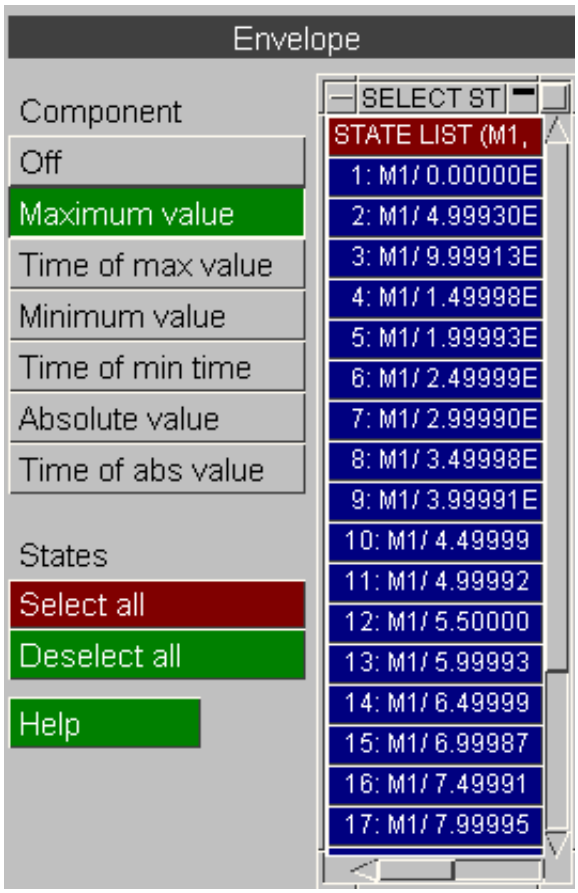
Note 1: While thick shells write results at surfaces too, this flag has no effect when they are plotted since their (visual) thickness permits all three surface results to be displayed simultaneously on their respective faces.

Note 2: The **Shell Surface** figure above is exceptional in that it also demonstrates **Layer** selection for shells. You will not normally see this: it is the result of a non-default output option in LS-DYNA, intended for composites analysis, which writes results at multiple integration points. This is discussed further in Section 11.8.2.2.

Note 3: Historically LS-DYNA has reverted to trapezoidal integration for 6+ integration points, although this is undocumented and the author has a sneaking suspicion that more recent versions of LS-DYNA may use gaussian integration for up to 11 points. Examine such results with care!

A more detailed description of shell output, with particular reference to "surfaces", "layers" and integration schemes, is given in Section 9.8.

4.2.4.3 ENVELOPE...



Envelope plotting can be used to plot either the minimum, maximum or the absolute maximum data values within a range of states.

In addition to plotting the minimum or maximum values the times of the states that the values occurred at can also be plotted.

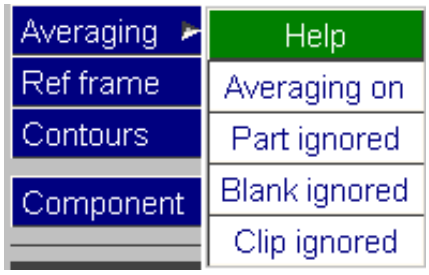
If envelope plotting is turned on for an element based data component (i.e strain) then element averaging is automatically turned off (see Section 4.2.4.6).

When envelope plotting is turned on Line Contours (see Section 4.2.4.8) are only available for node based data components (i.e velocity).

Envelope plotting may also be used with the **WRITE** option (see Section 6.7) but it is not available during **ANIMATION** (see Section 4.4.1) or when the **REFERENCE_STATE** option (see Section 6.3.5.1) is being used.

Note: At present **ENVELOPE** plotting only functions in 2D mode. Users running in 3D under OpenGL will be temporarily swapped back to 2D mode for the duration of an envelope plotting operation.

4.2.4.4 AVERAGING... Controlling data averaging across adjacent elements.



By default data for contour plots is averaged across adjacent elements of the same type, regardless of their material, etc.

These settings allow you to modify this behaviour.

The averaging control settings have the following meanings:

Averaging Switches averaging on or off, (default on). If turned off then no data averaging occurs for contour plots: this results in a "patchwork quilt" effect for area contour plots, and no results at all for line contour plots.

Material ignored By default this is on, and the material type is not considered when averaging. If you switch this off then averaging will **not** take place across adjacent elements if they are of different materials.

Blanking ignored By default this is on, and blanked elements are still included in the averaging process even though they are not visible. This means that blanking elements will not change the values used for contouring on those that remain. If you switch this off then results from blanked elements are **not** included when results are averaged at nodes.

Clipping ignored By default this is on, and elements that have been volume-clipped from the display are still included in the averaging process even though they are not visible. If you switch this off then results from volume-clipped elements are **not** included when results are averaged at nodes (ie same logic as is applied to blanked elements above).

Notes on data averaging:

Note 1: Averaging of element data at nodes for contouring only takes place for element derived data, eg stresses. Where the data being plotted is nodally derived, eg velocity, then averaging is not used and the settings above have no effect.

Note 2: Averaging has an effect beyond plotting: it can also influence how element-derived scalar data is computed at nodes for **WRITE** and **XY_DATA** output.

Note 3: Averaging never takes place over dissimilar element types. For example where a node is common to both a solid and a shell data at the node is computed separately for the "parent" element types, even if the data component type is valid for both types.

Note 4: Averaging is applied if requested, even if it might not be sensible to do so: this can be an issue when directional components are plotted in element local systems. For example if **LOCAL X_DIRECT_STRESS** is used where two shells meet at a right angle (ie a flange meets a web) you may be averaging stresses in directions that are 90 degrees apart.

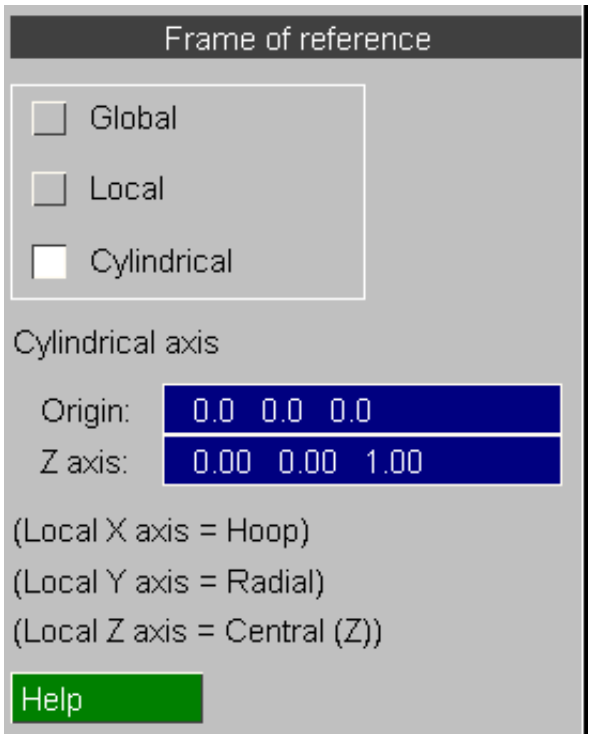
Note 5: A related error is to average across shells, using top or bottom surface data, when adjacent shells have inverted surfaces; ie their outward normals (local Z axes) point in opposite directions. This is usually the result of a meshing error, and it can produce strange contours. (To check outward normals turn on the element local triads with **DISPLAY_OPTIONS**, **LOCAL_TRIAD;** or do a continuous-tone (**CT**) plot of the element outward normals using (geometric) component **ON_OUTWARD_NORMAL**.

4.2.4.5 **REF_FRAME...** Choosing the frame of reference.

Directional data components may be plotted in model **Global,** element **Local** or (global) **Cylindrical** coordinate systems. The default is **Global**.

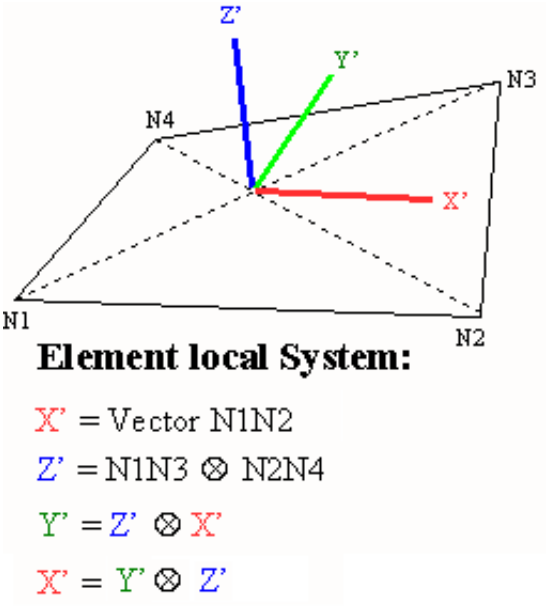
Directional components are basic stresses and strains from their respective tensors, ie **X_DIRECT_STRESS**, **SX_DIRECT_STRAIN** etc. See Section 9 for more details.

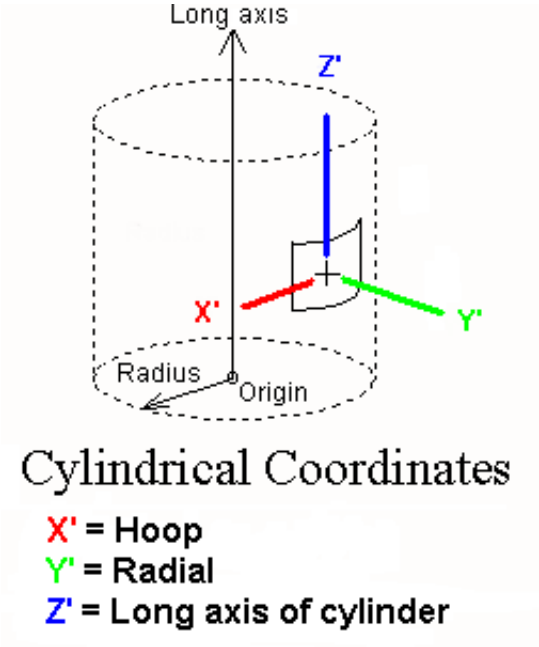
In this example a **Cylindrical** system has been used, and the user has defined the origin and vector of the local Z' axis.



The element local system is computed from its topology. A four noded element is shown here, for a 3 noded element Z' is normal to the (flat) plane N1N2N3.

A cylindrical system is only suitable for elements that do genuinely lie in the plane of a cylindrical wall. The Y' vector is perpendicular to the Z' axis through the element centre.





Element local axes are calculated as follows:

[**X'**] (approx)	From vector **N1N2**
[**Z'**] (outward normal)	From cross product **N1N3** x **N2N4**
[**Y'**]	From cross product [**Z'**] x [**X'**]
[**X'**] (warping correction)	From cross product [**Y'**] x [**Z'**]

Let [**U**] be the vector from the cylinder origin to the element centre, then axes are calculated as follows:

[**X'**] (hoop)	from cross product [**U**] x [**Long axis**]
[**Y'**] (radial)	from cross product [**Long axis**] x [**X'**]
[**Z'**]	is the same as the [**Long axis**]

The data to which coordinate system transformations are applied

- **Element tensor derived data** is transformed to Element Local or Cylindrical.

  "Tensor derived" means Stress and Strain tensors, and any User-defined tensor components.

- **Nodal vector derived data** is transformed to Cylindrical only. ("Local" has no meaning for nodes)

  "Vector derived" means Displacement, Velocity and Accelerations vectors, and any User-defined vector components.

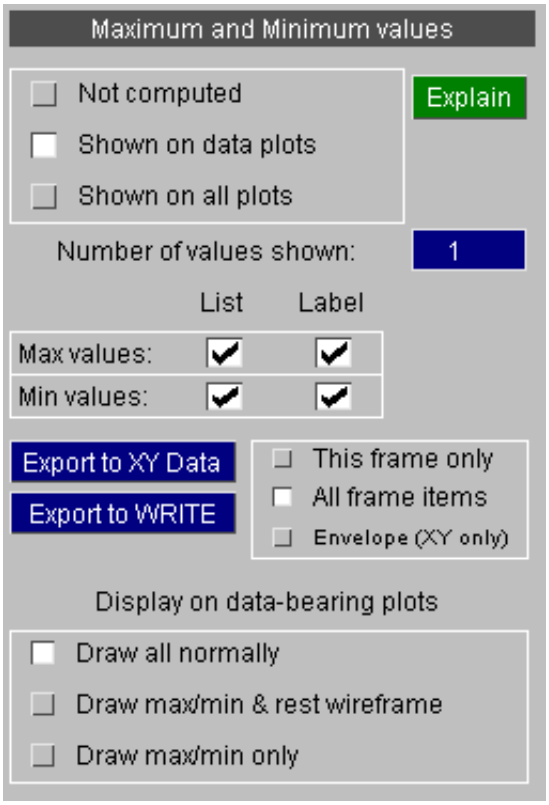
  One exception is that vector plots of nodal vector data are always presented in the global system in order to show the "true" vector directions.

4.2.4.6 **CONTOURS...** Setting contouring attributes.

There are many attributes of contouring that can be set. This panel is the "generic" level giving access to specific settings. It also shows the current settings of major parameters.

Since this panel has any sub-options and is common to all plotting modes so, to avoid repetition, it is described separately in Section 4.3.

4.2.4.7 **MAX & MIN** Displaying max and min values on plots



D3PLOT can calculate the maximum and minimum <n> values on the current plot, and display it at the top of the graphics window. Element-derived data components show element values, and nodally-derived ones show nodes.

Not computed	No max/min values computed or displayed
Shown on data plots	Max/min values are computed, but only displayed on data bearing plots (CT, SI, etc)
Shown on all plots	Max/min values are shown on all plot types. For non data-bearing ones the values shown are those of the currently selected data component.
Number of values shown	By default only 1 of each max and min is shown. You can choose any number, but space on the plot is limited and a practical limit is about 30 pairs of values. Very large numbers will also take longer to compute and store.
List/label options	By default values are both listed on the plot, and the relevant elements/nodes are labelled to identify them. Both are switchable.

Export to XY Data

This option will calculate the max and min values for all states in the analysis, and export them as (x,y) data to the XY_DATA tool for graphical plotting. The actual results sent depend on the settings below:

This frame only.	Generates curves only for the <n> items that are the max and min in this frame. This will result in <2n> curves.
All frame items.	Generates curves for the <n> items that are the max/min in each state. This can result in up to 2 x #states x <n> curves if the max/min items in each state are different.
Envelope	This produces only two curves, a maximum and an minimum. The X axis is state time, and the Y axis is the actual max/min value at each state regardless of the actual element/node it comes from.

Export to WRITE

This option is similar to **Export to XY Data**, except that it builds the list of elements/nodes based on the options selected, and exports them to the **WRITE** tool for numerical output at the current time.

Display on Data-bearing plots:

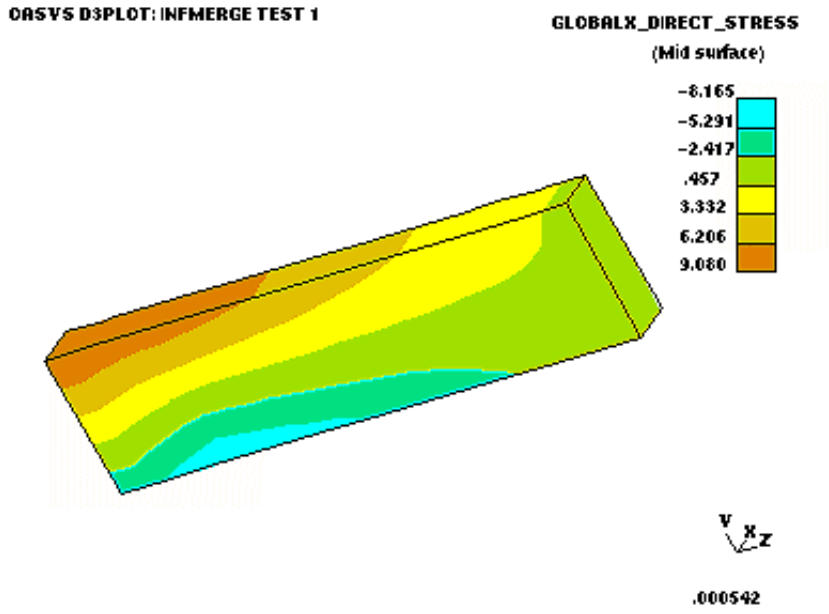
Draw all normally	Normal plotting with no restrictions
Draw max/min and rest wireframe	The max and min <n> items are drawn in the current plotting mode, and the rest in wireframe mode
Draw max/min only	Only the <n> max/min items are drawn in the current plotting mode.

4.2.4.8 Plotting commands for Solids and Shells.

There are six display modes for plotting data on solid and shell elements. (In all the examples below a free-edge element overlay has been used to simplify plots).

CT (**CT_CONTINOUS_TONE**)

A continuous-tone plot draws bands of contours in solid colours. No account is taken of lighting.

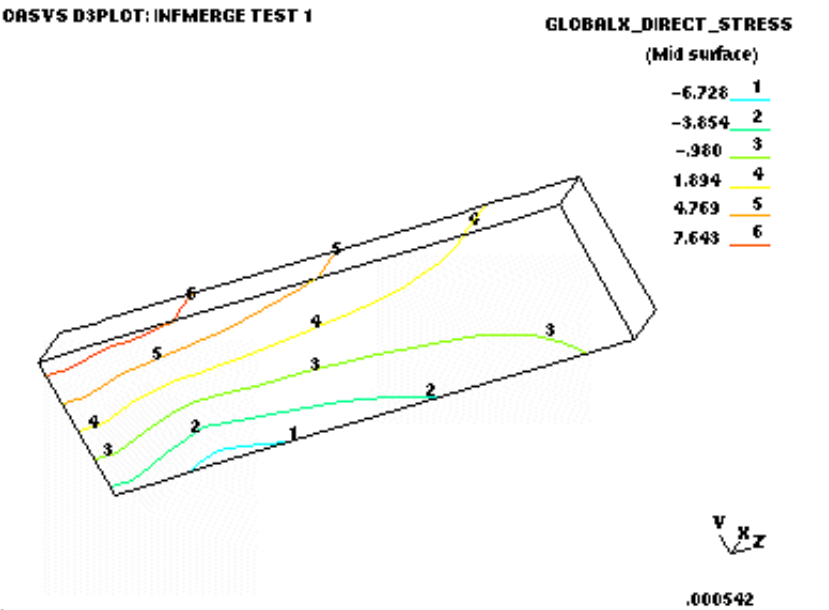


LC (**LC_LINE_CONTOURS**)
A line-contour plot draws lines of constant value across elements.

In this example the lines have been labelled with their contour band numbers: this is optional, and controllable using the **CONTOUR** command.

Note that the contour values, and hence the lines, lie in the middle of the equivalent solid bands above.

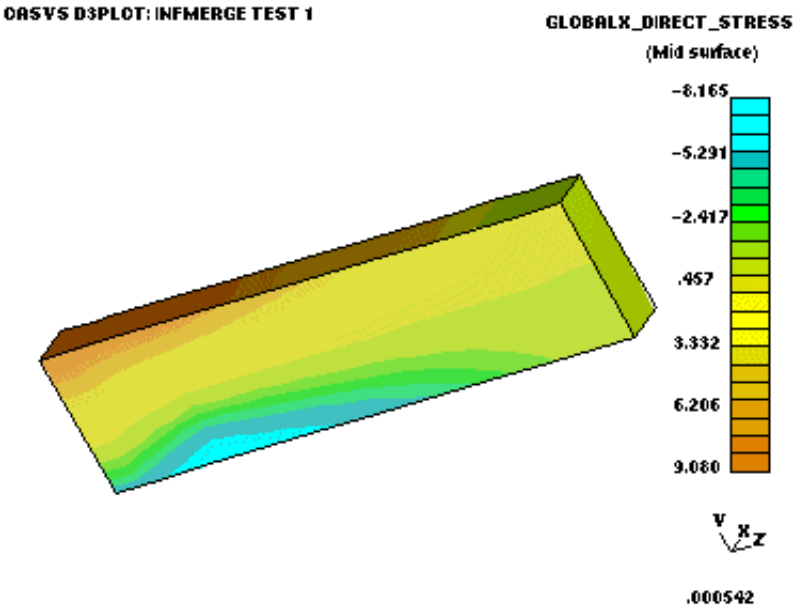
By default **LC** mode plots are drawn as lines on a hidden-line element mesh, but "mixed-mode" plotting allows them to be superimposed on shaded or wireframe meshes too. See the **PROPS** box in Section 4.2.2.4.



SI (**SHADED_IMAGE**)
This is similar to a continuous-tone plot, but lighting is included to give more impression of shape.

The **LIGHT** command, see Section 4.2.3, controls the lighting aspects of the plot; and there are special options in **CONTOUR**, see Section 4.3, to control the contouring aspects.

Two options are provided:



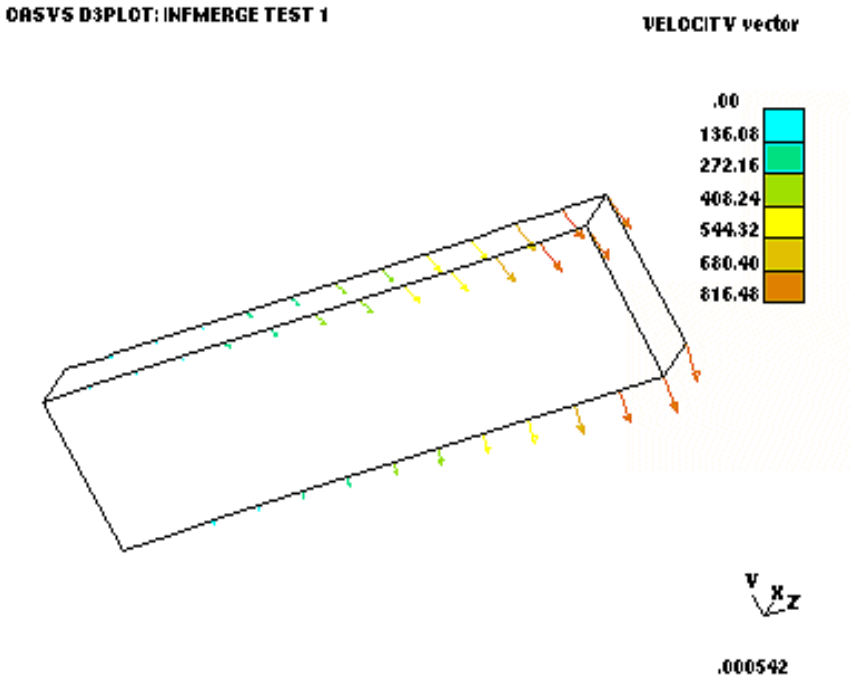
Solid Bands (Default) Draws solid contour bands, exactly as in a **CT** plot, except that lighting is added.

Fuzzy bands Draws smoothly varied colours, gouraud shading, with lighting. (This was the standard SI mode prior to release 9.2)

VEL (**VELOCITY_PLOT**)
A velocity plot draws arrows showing the direction and magnitude of nodal velocities.

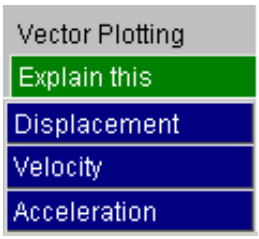
Arrow length is proportional to velocity magnitude, and arrow colour is also based on magnitude using the normal contour band colours and bands.

Both of these attributes are set with the **CONTOUR** command - see Section 4.3.



Vector plotting can select any of **Displacement**, **Velocity** and **Acceleration**; the default being velocity.

If user-defined vector components for nodes have been defined they are also eligible for plotting in this mode.



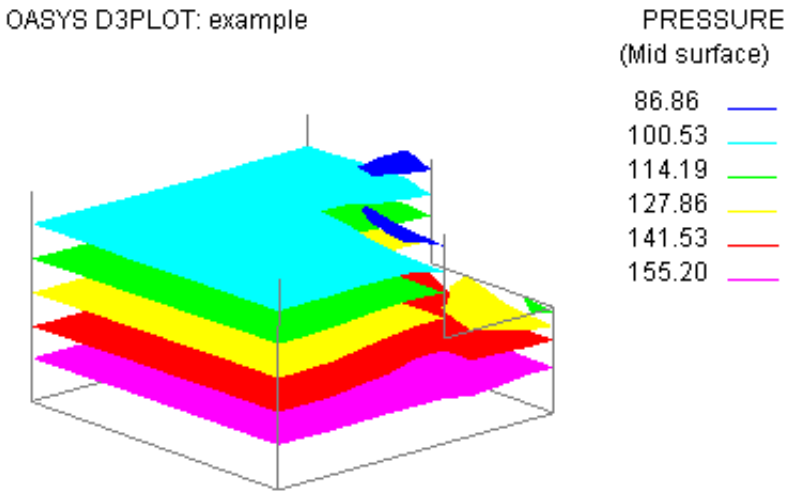
Like LC mode plots **VEL**ocity plots are drawn on a hidden-line mesh by default. However mixed-mode plotting means that the underlying mesh may be rendered in shaded and/or wireframe modes - see the **PROPS** box in Section 4.2.2.4.

ISO
ISO surface plots draws surfaces at constant values, with lighting added.

In 3D elements, as shown here, the contours will show surfaces of constant value through the body of the solid mass.

In 2D elements the effect is the same as a Line Contour plot, drawing lines of constant value.

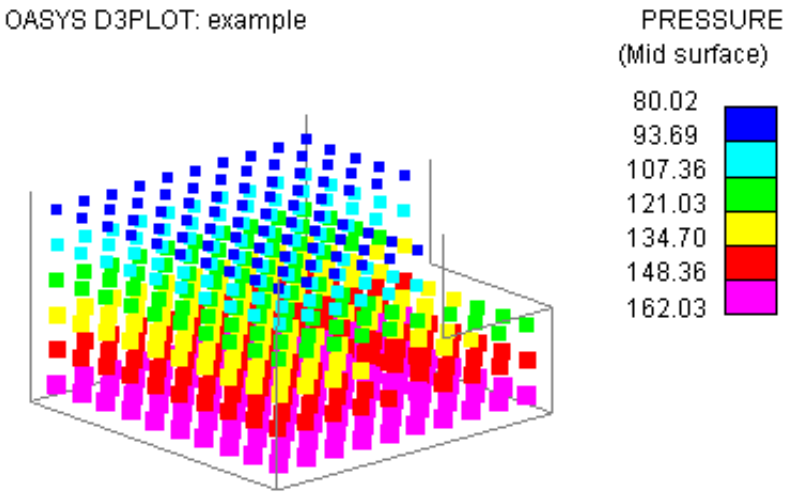
Note that ISO plots of large solid meshes can be many times slower than, for example, SI plots of the outer surface. This is due to the "cube /square" law: an ISO plot has to consider all elements inside the mesh, whereas an SI plot need only consider external elements.



CL (**CLOUD_PLOT**)

A cloud plot produces points displaying the value of the selected data component at each node. These can been drawn as a fixed size of the user's choice or proportionally to their value. This property can be set with the **CONTOUR** command - see Section 4.3.

CL plots are a far "cheaper" way of seeing what is going on inside large solid meshes than ISO plots (or cut sections). This is because they don't have to worry about data averaging or complicated graphics, making them much faster to process.



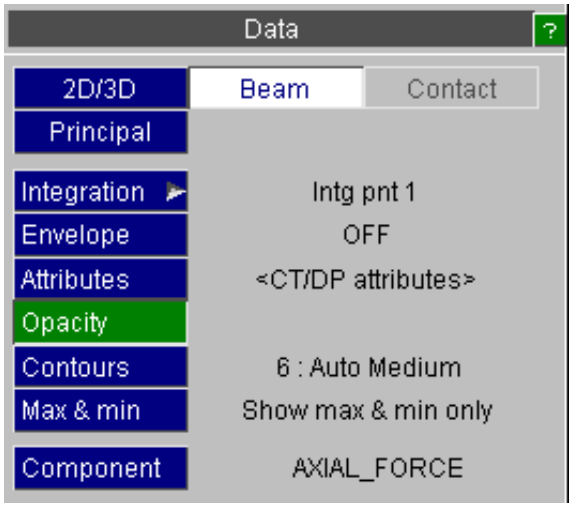
This section has described the commands in the **2D/3D Solids and Shells** plotting mode panel. But you should be aware that there are further commands that can influence the appearance of all plots located in the top menu box:

Display_options, **Label**, **Entity**, **Blank**, **Volume_Clip**

These are described in Sections 6.1 et seq.

Click here for the next section

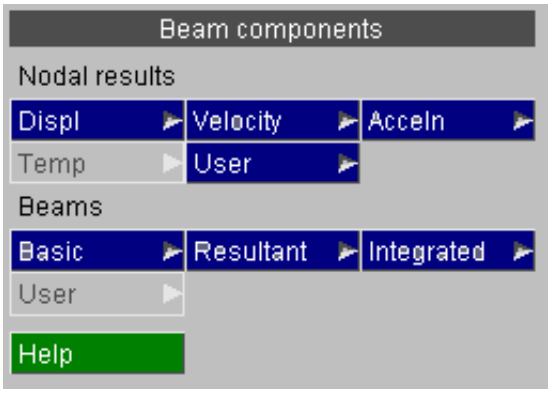
4.2.5 Beam Plots Plotting results on beam elements



Beam elements cannot be contoured in the same way as 2 and 3D elements since they have no visual area on which to plot results.

Therefore the special "Beam Plotting" data display mode must be used to visualise results from beams.
Note: Results from other 1D elements: springs, dampers, seat-belts, joints, etc are not currently written to database files for plotting, and they cannot be displayed in this (or any other) D3PLOT menu.

4.2.5.1 COMPONENT... Selecting a beam data component



All beam types have the **BASIC** set of data components: 3 forces + 3 moments.

Belytschko-Schwer beams using a resultant plastic formulation will have "extra" **PLASTIC** components.

Hughes-Liu (integrated) beams can optionally write "extra" **STRESS** (and strain) values at integration points.

D3PLOT supports both basic and "extra" data components for all these beam types.

However note that LS-DYNA contains many different beam formulations, and it is not possible to tell from the database files what the data for a given beam actually mean. It is your responsibility to interpret your results correctly, and you should read the notes in Section 12.10 before processing beam data.

4.2.5.2 INTG_POINT...

Selecting the integration point for results from "integrated" (Hughes-Liu etc) beams.

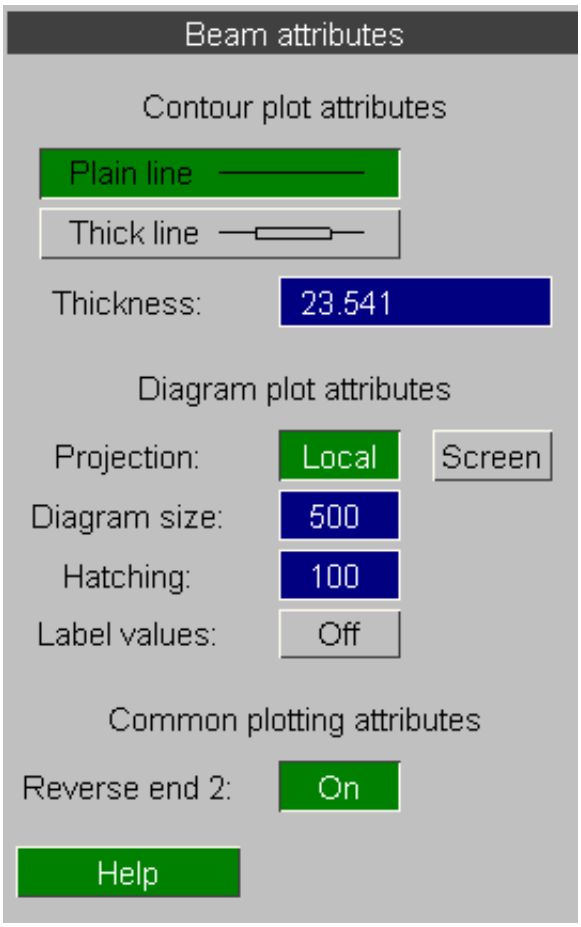
The "extra" **INTEGRATED** results for these beam types only are written at the specified integration points. Only one can be plotted at a time, and this is selected here.

(This is analagous to defining the integration point for output of shell data.)

4.2.5.3 ENVELOPE...

Envelope plotting for beams works in exactly the same way as for other element classes. The selection of states, data components and so on are identical. (See Section 4.2.4.4.)

4.2.5.4 ATTRIBUTES Setting the attributes of beam plots



Beam results can be plotted in Continuous-tone and Diagram-plot modes. These have special visual attributes that are set in this panel.

The examples below all use a simple built-in beam with a point load at its centre, and show the resulting bending-moment diagram as affected by the settings here.

The "thick" line used will depend upon whether or not a .ZTF file has been read in from Oasys Primer.

- If it has been read then the beam's "true" section thickness will be used.
- If no such file has been read a square section of the "Thickness" given here will be used.

(D3PLOT V92 can reproduce a range of "true" section shapes from the *SECTION_BEAM data encoded in the .ZTF file.

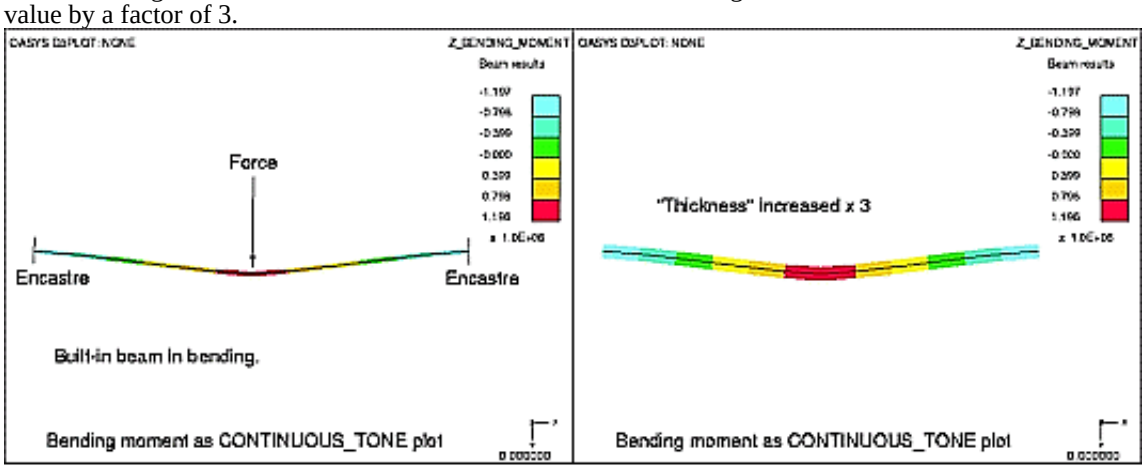
"Integrated" beams with explicit section shapes are drawn as specified.

"Resultant" beams with section properties (A, Ixx, Iyy) have a hollow rectangular or tubular sections inferred from their properties.

Beam symbol display is discussed further under Display Options

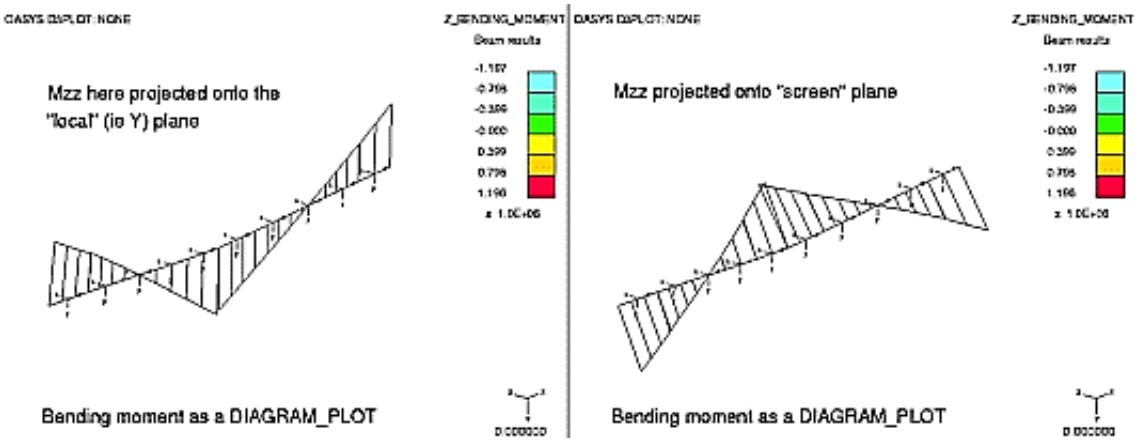
THICKNESS Setting continuous-tone plot image thickness.
Continuous-tone beam plots show the data values on a beam as thick bars of solid colour along the beam centre-line. The thickness of these bars is set by default to 1% of the model's longest dimension (in model space units), which proves adequate for most models.

You can change this value with the **THICKNESS** command. The figure above demonstrates the effect of changing this value by a factor of 3.



PROJECTION Using "local" or "screen" projection for diagram plots.

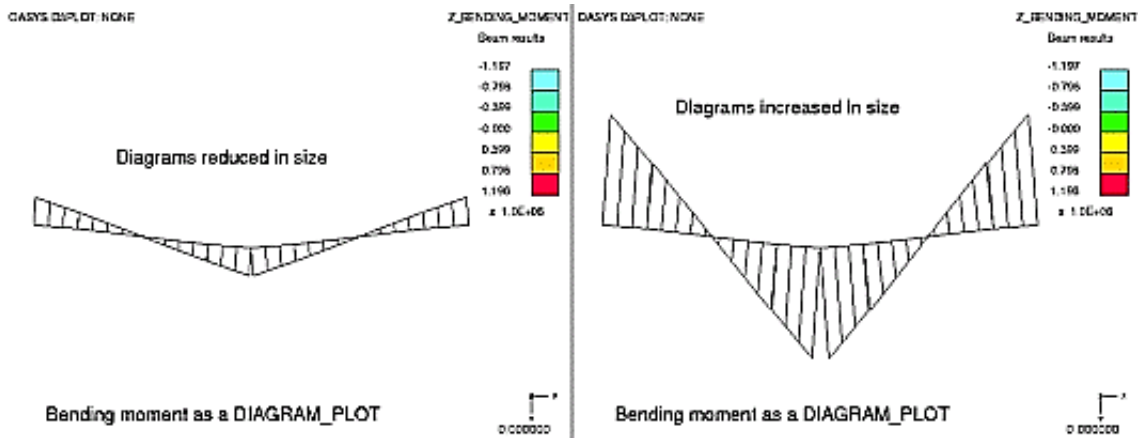
By default those data components which imply a local direction (ie shear and bending in/about beam local Y and Z axes) are drawn in "diagram" plots projected onto their "local" plane. For example in the left hand side of the figure above the bending moment **Mzz** is shown projected onto the plane of beam local XY. (The beam local axes have been turned on for clarity.) This gives a visual indication of the direction and sign of the component.


Mzz here projected onto the "local" (ie Y) plane
Bending moment as a DIAGRAM_PLOT
Mzz projected onto "screen" plane
Bending moment as a DIAGRAM_PLOT

You can choose instead to project results onto the "screen" plane, as shown in the right hand side of the figure above. This draws results in the plane of the screen regardless of the beam's orientation. This mode of display is used unconditionally for components that do not have an implicit direction (eg axial force, torsion, etc)

DIAGRAM SIZE Setting the visual scale of diagram plots.

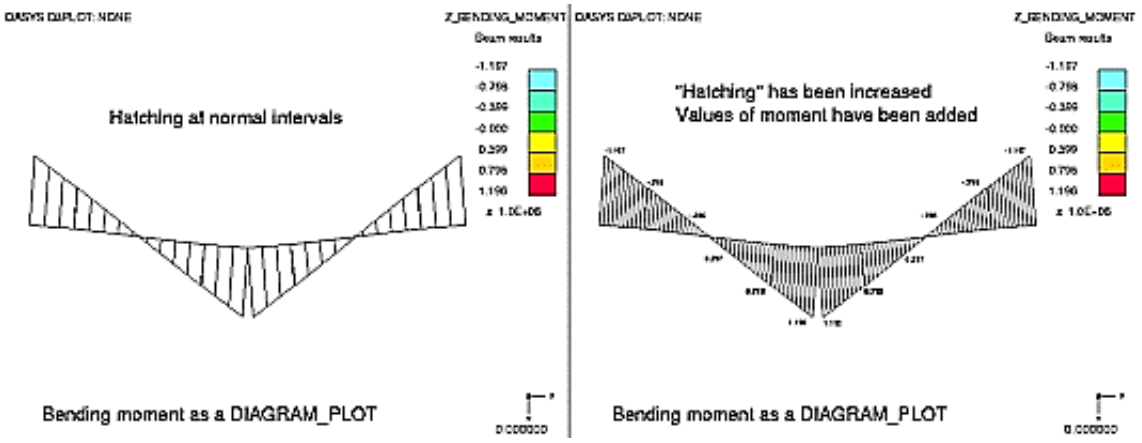
The size of diagram plots is set by default such that the largest vector is about 500 screen units. You can change this at will: the figure above shows typical settings.



HATCHING Setting the density of diagram plot hatching

LABEL_VALUES Labelling values on diagram plots.

The left hand figure above shows the default hatching density: at roughly every 100 screen space units. On the right the hatching density has been increased by **reducing** its spacing to every 30 units.


Hatching at normal intervals
Bending moment as a DIAGRAM_PLOT
"Hatching" has been increased
Values of moment have been added
Bending moment as a DIAGRAM_PLOT

Note: A "solid" diagram plot appearance can be achieved by reducing the hatching spacing to typically 10 units or below. The actual value will depend on the screen window size and device resolution. It requires a lot of screen vectors to achieve this effect, so don't use it where display speed or storage space are critical: eg during animation.

The right hand figure above also shows the effect of turning on the **LABEL_VALUES** switch: values at beam end points are shown. This option is only useful when there are relatively few beams on the display, with a lot of beams being labelled the screen will become a mass of numbers. (The **LABEL** option, using "dynamic" labelling of beams with the **DATA_VALUE** switch on, provides a more selective way of drawing element data values on the screen.)

REVERSE_END_2 Sign convention used for beam plot display.

The convention when drawing bending moment diagrams is to plot the diagram on the tensile side of the element. However the mathematics would suggest otherwise:

In the example used here (encastre beam, point load) the moments at the two ends of this beam are clockwise (+ve) and anti-clockwise (-ve) respectively, so at one end a +ve value must be drawn on the tensile side but at the other end a -ve value. Hence the need to reverse the sign of the "end 2" value for plotting purposes only. The author has never seen plots drawn without this reversal but, for completeness, the ability to produce them is provided with the **REVERSE_END_2** switch: turn it off to see mathematically "pure" plots.

4.2.5.5 **WARNINGS** On-line warnings about beam-plotting pitfalls

There are several different beam formulations in LS-DYNA, the principal ones being:

Hughes-Liu	Standard/arbitrary section, integrated at mid-span
Belytschko-Schwer	Standard sections, integrated at two ends
Truss	Axial only Belytschko-Schwer beam
Discrete	Zero-length, generalised spring-like behaviour
Cable	Tension only
Spotweld	A variation on discrete

Various data output options exist depending on material type, beam formulation, number of integration points and user-defined settings. Unfortunately the output files do not contain enough information to allow beam results to be diagnosed unambiguously, so **WARNINGS** provides on-line guidance. You should also read Section 12.10 which describes beam output.

Inconsistent sign conventions in LS-DYNA releases up to and including 970

Due to a bug in LS-DYNA versions up to and including LS970 exhibit the following inconsistent sign convention for beam output:

- "Resultant" (typically Belytschko-Schwer) elements use one sign convention
- "Integrated" (typically Hughes-Liu) elements use the opposite sign convention for 4 of the 6 output components.

The following table shows the results from releases 970 and earlier:

Component	Matching?
Fx	Same
Fy	Opposite
Fz	Opposite
Mxx	Opposite
Myy	Opposite
Mzz	Same

Which is right?

Sadly there is no "right" for beam output, as different users have different conventions. The confusion arises because of the different ways in which the beam types work: integrated beams have integration points at their centre, whereas resultant beams have (potential) hinges at their ends. The former reports force in the beam, and the latter reactions at the supports.

D3PLOT attempts to draw bending moment diagrams on the tensile side, but depending on which beam type you have used this may or may not be the case.

Sign conventions are consistent from LS-DYNA release 971 onwards

At some stage during the development of LS971 this problem was fixed, and results now use the "integrated" convention for all beam types.

Unfortunately D3PLOT cannot tell with certainty which LS-DYNA version was used to generate a set of results, so it cannot correct for this automatically. This has consequences for cut-section force extraction from beams, described in section 6.4.4

Interpreting beam results requires knowledge about the original beam and material formulations that is not available in the database (`.ptf`) files. As a consequence there are some ambiguities in the interpretation of results, especially for "extra" data, that D3PLOT is unable to resolve for you.
It is your responsibility to determine what your results mean,
and to interpret them correctly.

If you need advice or help please contact Oasys.

4.2.5.6 **OPACITY_SWITCH** Making overlying structure transparent

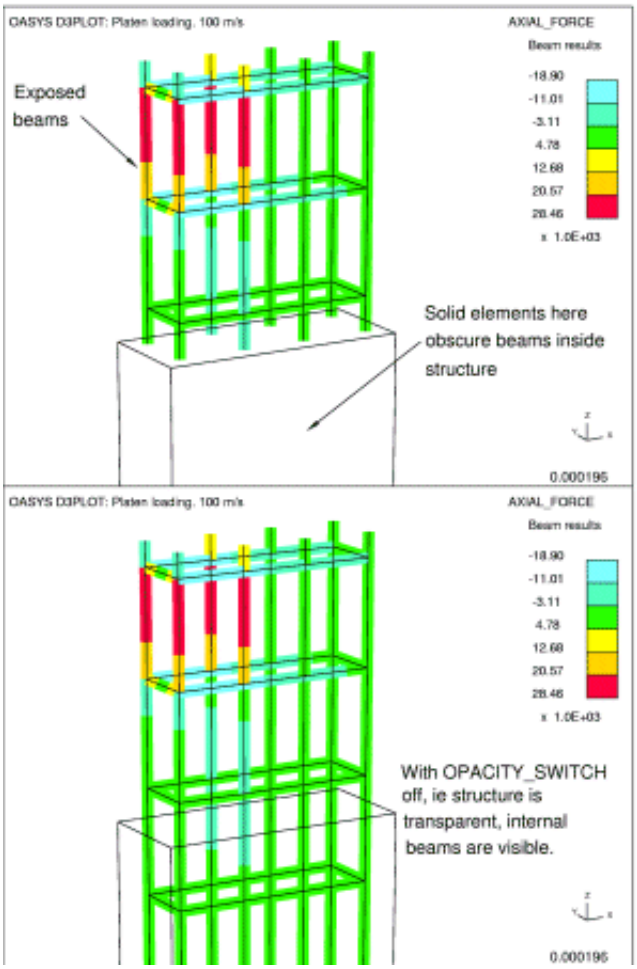
Beams are often buried inside a model which also contains shells and solids, and it can be difficult to see beam data plots because of the intervening structure. Therefore D3PLOT allows you to make this overlying structure transparent when performing beam plots.

This option is preserved for backwards compatibility. A more flexible method would be to adjust the visual transparency of the overlying structure, using the "quick pick" option, or explicit settings in the **PROPs** panel.

Normally overlying structure will obscure (correctly) any beams that are behind it.

This example shows a typical concrete column with reinforcement and, clearly, it would be useful to visualise forces in the enclosed area while still seeing the external concrete outline.

When the **OPACITY_SWITCH** is turned off the overlying structure is no longer opaque, (ie it becomes transparent), and the results in the beams obscured above become visible.


Exposed beams
Solid elements here obscure beams inside structure
With OPACITY_SWITCH off, ie structure is transparent, internal beams are visible.

By default the **OPACITY_SWITCH** is off, ie beams which should not be visible are indeed obscured by the overlying structure. It can be turned on/off at will.

Note: The **OPACITY_SWITCH** affects both beam and contact surface data plotting modes (the same switch in both contexts). It has no effect in other contexts, or upon plotting modes that do not display data.

4.2.5.7 Data plotting commands for Beams.

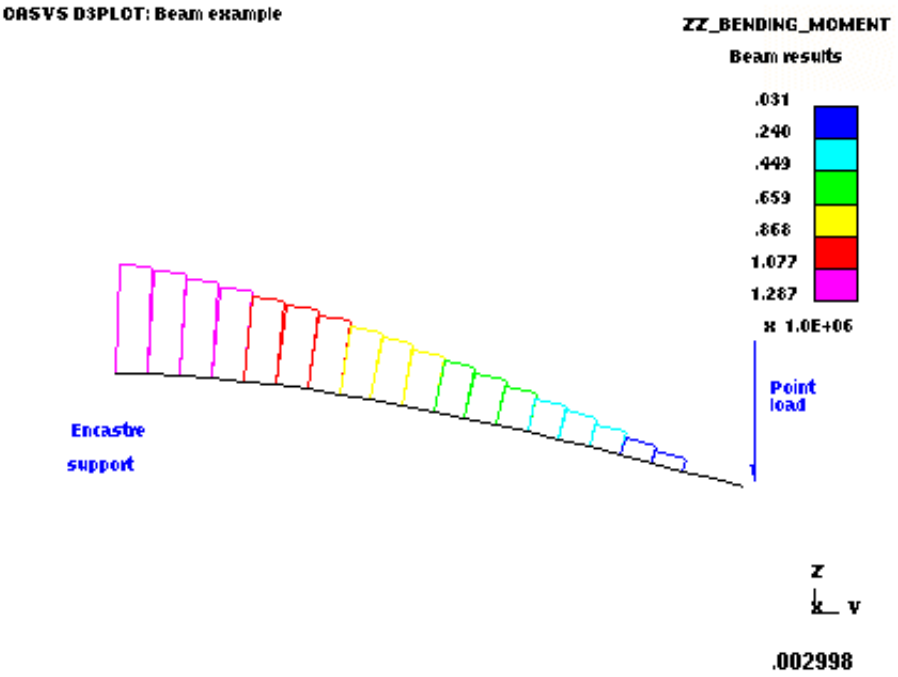
There are two data plotting modes for the display of beam results.

DP (**DIAGRAM_PLOT**)

This shows the **DP** option: a "diagram" plot showing results hatched on the beam.

Hatching size is proportional to data magnitude, and colour also follows the normal contour band limits. Directional data is displayed by default on the relevant local beam axis.

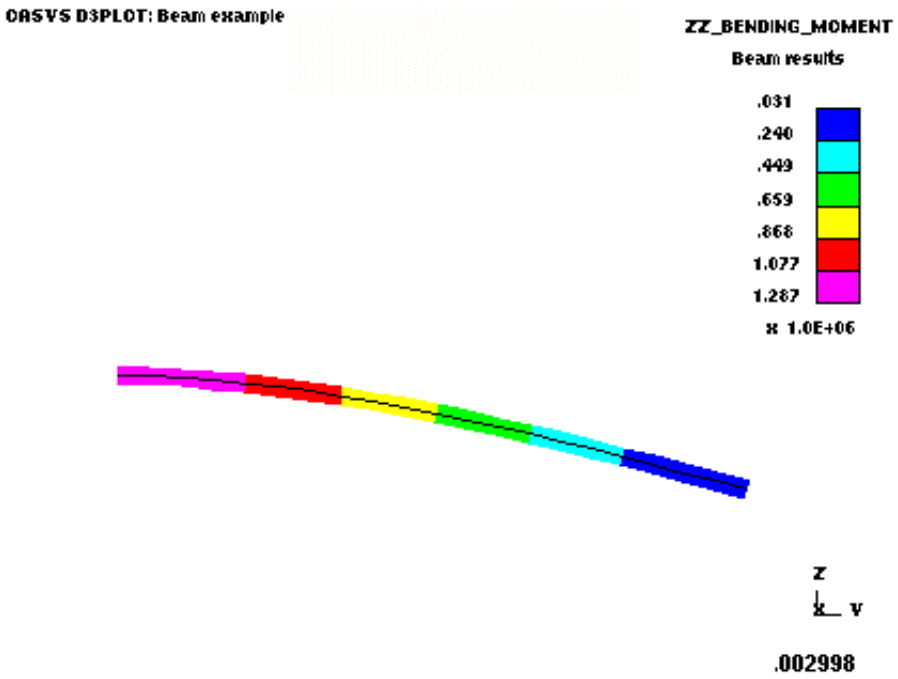
Any data component can be displayed this way, not just bending moments.



CT (**CONTINUOUS_TONE**)

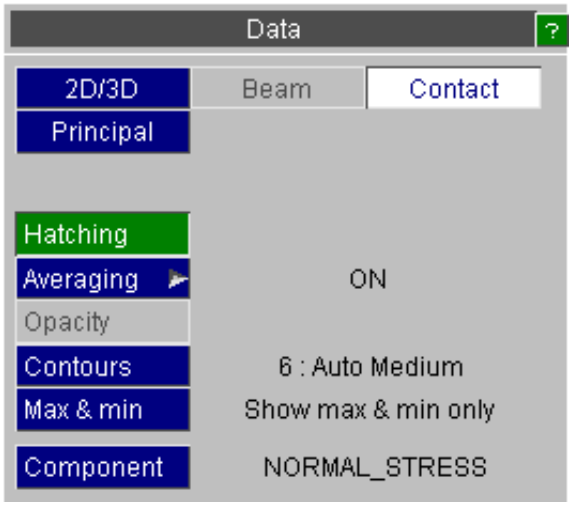
This **CT** continuous-tone plot shows exactly the same results as in the diagram plot above. Now the results are drawn as thick blocks of colour on the beam centreline.

This can give less cluttered plots, but no visual indication of the data or beam orientation is possible.



Click here for the next section

4.2.6 Contact Surface Plots Plotting results on contact surface (interface) segments



Visually contact surface segments have the same attributes as thin shells, and data may be contoured on them in the same way. To distinguish between the two element types contact segments in D3PLOT are hatched by default.

This contact surface plotting mode must be used to display contact stress and force on segments. The distinction between this and normal 2D/3D element plotting is made to prevent confusion between the two element types.

Note: If you have not read in a .CTF file D3PLOT will not show any contact segments as being present.

The following data components may be plotted:
The **STRESS...** and **FORCE...** categories are described in more detail in Section 12.11.

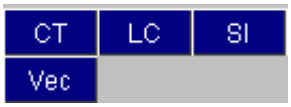
The **GEOMETRIC...** category allows segment outward normals to be plotted.

Contact surfaces generate penalty forces at nodes, which are converted to "stresses" in LS-DYNA by dividing contact force by segment area. Strictly this is "contact pressure", rather than "stress", and you should not expect these values to be the same as the (genuine) stresses in the underlying elements.

The contact surface results are a sanpshot at a particular time. As contact forces are particularly noisy, there may be a degree of randomness to the data plotted.

The **OPACITY** switch contols structure transparency, allowing the contact forces to be seen through the structure.

Data plotting commands for contact surfaces

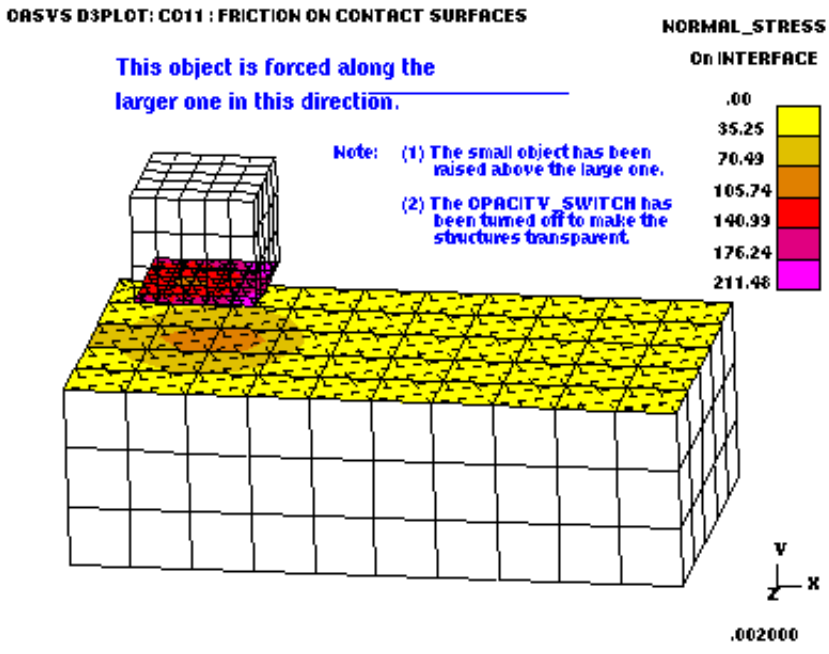


There are four plotting commands for the display of contact surface results:

CT (**CONTINUOUS_TONE**)

Continuous-tone plots of contact surface data are identical in concept to those of 2D/3D data: solid contour fill, no lighting effects.

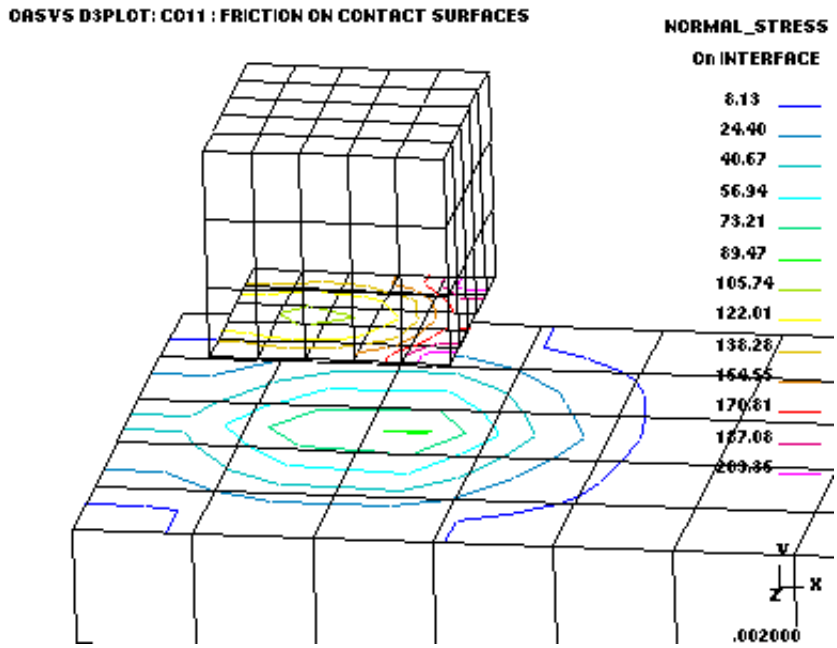
Note, however, that elements which are not contact segments are not contoured at all (as distinct from receiving zero values).



LC (**LINE_CONTOURS**)

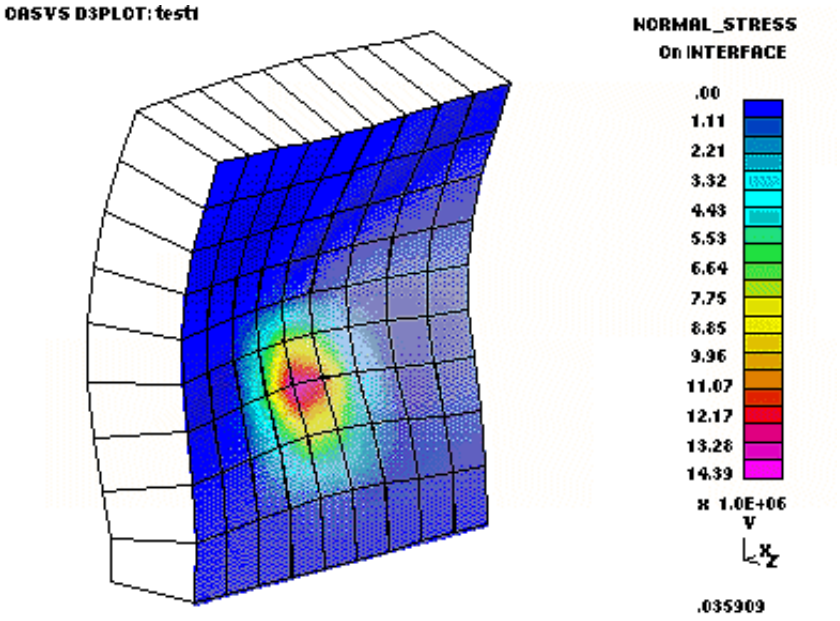
As with continuous-tone plots a Line-Contour plot of contact surface data is identical in concept to that of 2D/3D data.

(This is the same model as above, but with #contour levels increased to 13, cross-hatching turned off for clarity, and the scale increased to show detail.)



SI (**SHADED_IMAGE**)

The shaded-image plotting mode is identical in concept to that used for 2D/3D elements: 21 solid contours with lighting taken into account.

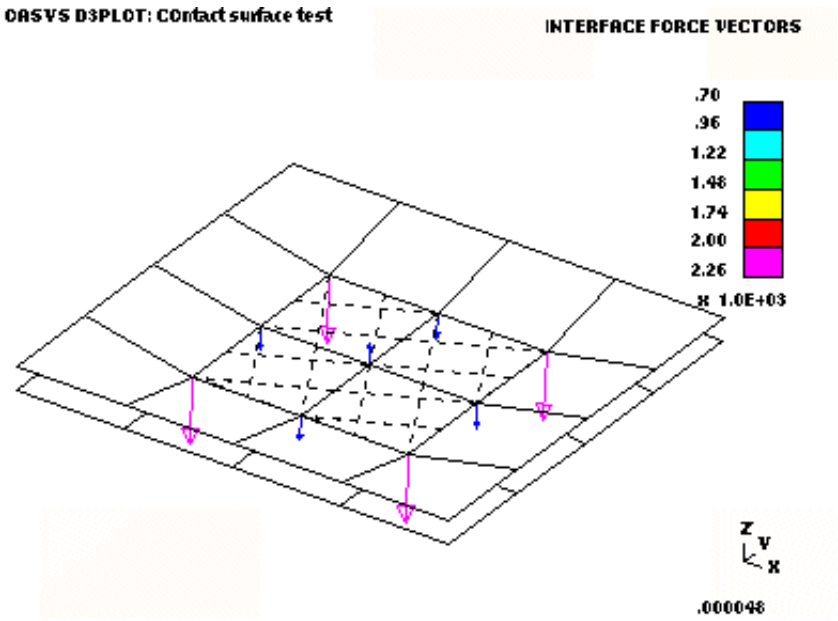


VEC (**VECTOR_PLOT**)

Vector plots draw arrows of force at nodes on contact surfaces.

Length is proportional to data magnitude, and colour is assigned using the current contour bar colours.

(This example shows two spot-welded plates being pulled apart. The nodes in the hatched area form a welded contact-surface.)



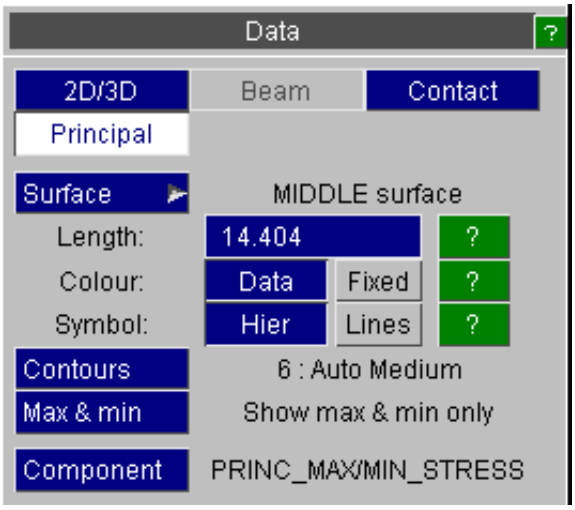
This section has described contact surface data plotting, and the (graphical) similarities with solid and shell (2D/3D) plotting in Section 4.2.4 will be apparent. As with that plotting mode there are further commands which influence the appearance of plots:

Display_options, **Label**, **Entity**, **Blank**, **Volume_Clip**

These are described in Sections 6.1 et seq.

Click here for the next section

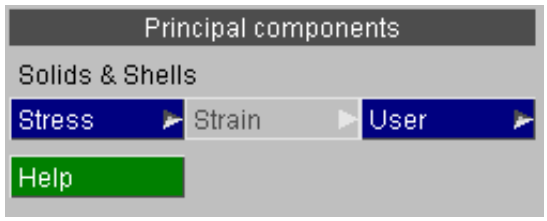
4.2.7 Principal Stress and Strain vector plots



This panel controls the plotting of principal stress and strain vector plots. (To contour these values use the normal 2D/3D

plotting commands described in Section 4.2.4)
These vectors can only be displayed on 2D and 3D elements (solids and shells), so if you have none of these in your model it will be unavailable.

4.2.7.1 COMPONENT Choosing a principal stress/strain data component



This panel shows the generic data component types for principal values. The **STRESS** and **STRAIN** categories are only made available if their respective tensors are present in the database file. See Section 9 for more details of element output and file contents.

Note 1: By default the strain tensor is not written by LS-DYNA, you must select it by setting **STRFLG** on the ***DATABASE_EXTENT_BINARY** control card. If your model seems not to have directional strain terms available this is probably the reason.

Note 2: As well as the simple (scalar) components it is possible to plot multiple values simultaneously. For example the component **PMM_PRINC_MAX_&_MIN** shown here plots vectors of both maximum and minimum stress, and **PALL** will plot all three principal values.

Note 3: If user-defined tensor components are available they too may be plotted using the components defined above. The transformation to principal is applied on the assumption that the user-defined tensor is in the global system.

4.2.7.2 SURFACE Selecting the thin shell intg point.

As with contouring 2D/3D data you must select the relevant thin shell surface for output. This is not necessary for solids and thick shells.
See Section 4.2.4.2 for more details.

4.2.7.3 **LENGTH**, **COLOUR**, **SYMBOL** Setting principal stress/strain vector plot attributes.

LENGTH: Setting vector length.

Vector symbols are scaled to data magnitude. **LENGTH** sets the maximum symbol length in model space units.

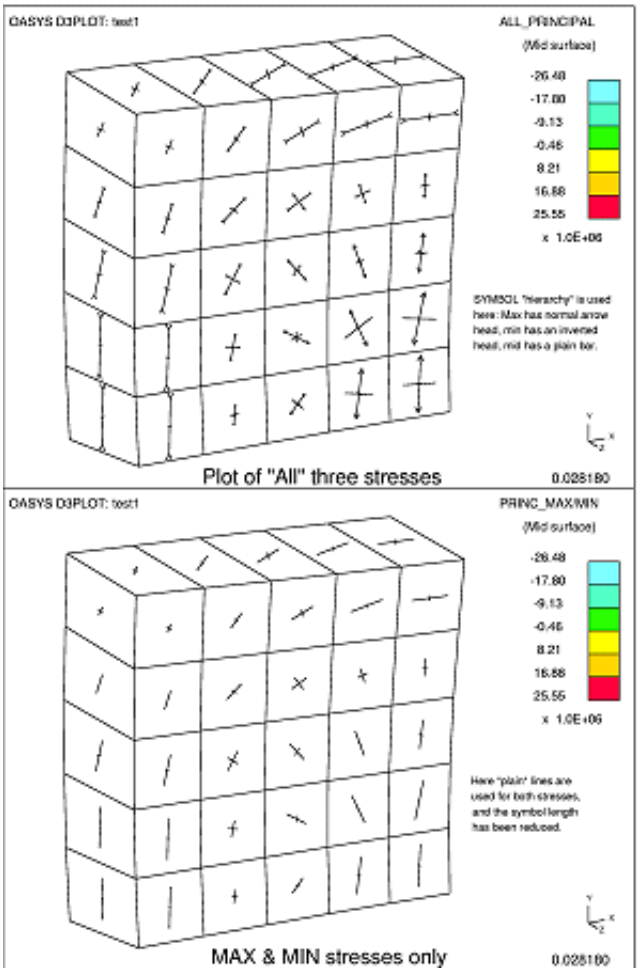
COLOUR: Setting vector colour
By default vectors are coloured using "contour" bands based on their data magnitude. This is the **DATA** option.

You can choose instead to use colour to distinguish between the components of multi-valued plots, the **FIXED** option.

SYMBOL: Vector symbol types

By default the vector symbols have a **HIER**archy: arrowhead for largest, inverted arrowhead for smallest, plain bar for middle.

You can choose to have plain **LINES** instead if you wish.



4.2.7.4 **GO** Executing principal stress/strain vector plots

The figure in Section 4.2.7.3 (above) shows examples of principal stress vector plots using the 3-valued component **PALL_PRINC_ALL** (all 3 principal stresses), and the 2-valued one **PMM_PRINC_MAX_&_MIN** (max and min principal stresses only). It also demonstrates the effects of the different **SYMBOL** settings. At present there is only this single data plotting mode.

Click here for the next section

4.3 Controlling contouring of data plots: **CONTOUR**

All of the data plotting commands described in Section 4.2 use the **CONTOUR** command panel to control data display.

The generic **CONTOUR** panel is shown right. The various options in it are described below.

Depending upon the plotting mode from which it is called, and the hardware being used, some options may be greyed out.

Click on the links here to jump straight to the relevant sections:

4.3.1: **Levels**

4.3.2: **Labels**

4.3.3: **Resolution**

4.3.4: **Arrows**

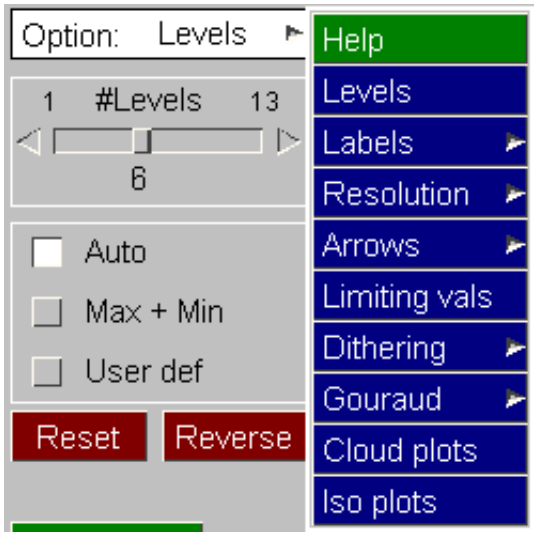
4.3.5: **Limiting values**

4.3.6: **Dithering**

4.3.7: **Gouraud**

4.3.8: **Cloud plots**

4.3.9: **Iso plots**



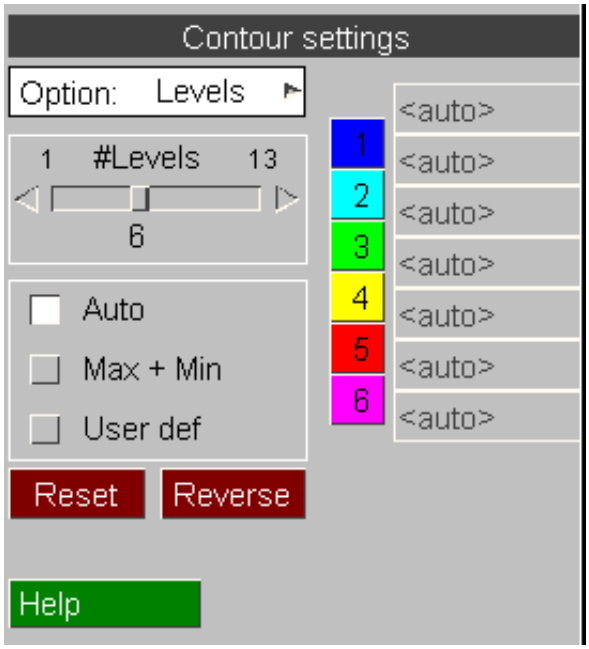
4.3.1 **LEVELS...** Setting number of contour levels, their ranges and colours

By default contouring is set to have:

- 6 levels
- Automatically computed values
- Colours from blue (low) to magenta (high)

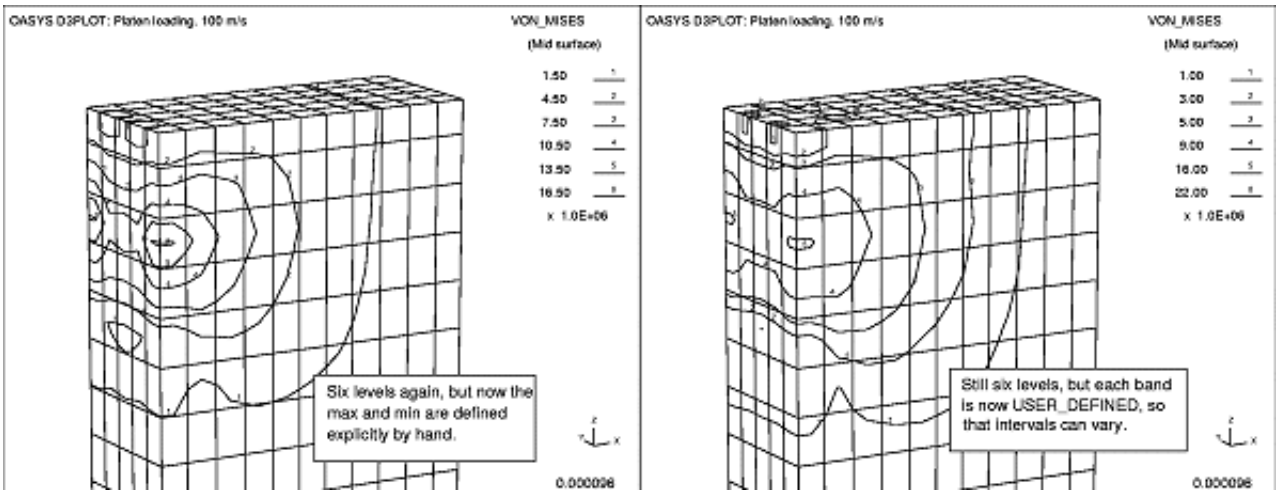
These default settings are shown right, the **LEVELS** sub-menu.

You can modify these settings as follows:



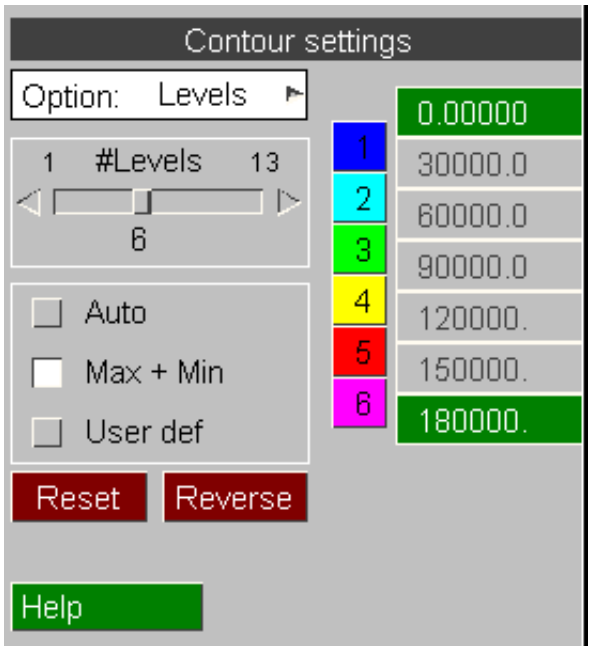
4.3.1.1 Setting contour levels. (**Automatic**, **Max_&_Min**, **User_Defined**)

By default contour levels are **AUTOMATIC**. This means that the maximum and minimum values are computed prior to each plot, and the resulting bands spaced evenly between these.



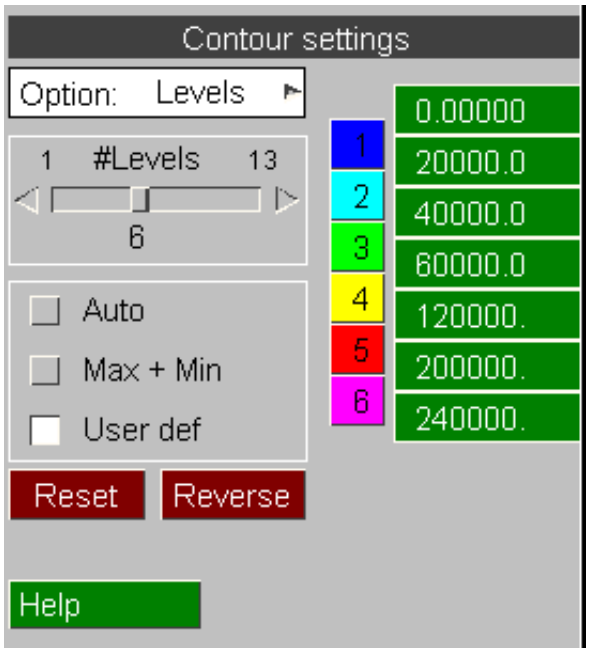
In the figures above the contour levels have been set manually: the max and min values only are set in the left figure, user-defined levels for each band are set in the right figure.

MAX_&_MIN levels have been selected, and the upper and lower values defined. The intermediate values are interpolated linearly and filled in for you, but you cannot change them.



The figure (right) shows the same panel set up for the plot in (b) above. **USER_DEF**ined levels have been chosen, and all contour bands filled in. Every level has to be defined: note the uneven intervals.

Hint: Choose **MAX_&_MIN** first, fill in upper and lower-bound values, then switch to **USER_DEFINED**. The interpolated intermediate values are remembered and may save typing.



Computing contour bands over multiple windows and models.

Where there is more than one window, possibly showing a different component, on a model; or more than one model in the database then contour levels are computed as follows:

AUTOMATIC contours:
For each active window (**Wn** tab selected):
- The max and min values of all visible elements in all models is computed
- These become the max/min bounds for that window
- Changing what is displayed in that window will update these bounds
- Each window is independently calculated, regardless of the contents of other windows.
- During animation exactly the same rules apply, except that the "envelope" of values from all frames is used to calculate the max & min values.

MAX & MIN contours:
For each active window (**Wn** tab selected):
- The user-defined max and min values are applied to that window, regardless of contents.

Applying the same contour bands to all windows.

It is often the case that you have several windows, and you want to have the same contour bands in all of them. To do this:

If you want to specify the bands to be used:
- Make sure that the **Wn** tabs for all required windows are selected.
- Choose **MAX_&_MIN** levels
- Set the required levels.

If you want to find the envelope of all windows, derive a max & min value from that, then set it in all windows.
- Make sure that the **Wn** tabs for all required windows are selected.
- Choose **AUTOMATIC** levels.
- Perform plots in all windows to update their local bands.
- If you want values over an animation then animate all these windows to force computation of the "envelope" max/min values in each.
- Then select **MAX_&_MIN**

The second method works because when you switch from **AUTOMATIC** to **MAX_&_MIN** D3PLOT computes the "envelope" of max/min values from all active windows, and applies this as the default values for **MAX_&_MIN** mode.

It is important to understand the distinction between automatic contouring over animation ***frames***, and automatic contouring at a given ***state***.

Automatic contour bands during Animation

During an ***animation***, or when a particular ***frame*** of an animation is displayed statically:
- For each window D3PLOT will scan all selected states and find the max and min contour values.
- If the window contains more than one model the max and min over all models in the window is found.

These max and min values become the contour bounds used for the animation, or when any frame of the animation is displayed, with the intention that contour bands in any frame will have the same values.

Commands which display animation ***frames*** are:
- [**>**] Play to initiate an animation, either at the top of a graphics window or in the "States" panel.
- Any of the **<< |< >| >>** frame positioning commands either at the top of a graphics window or in the "States" panel.
- Using the state slider at the top of a graphics window.
- Using the <shift> + <arrow key> short cut to toggle through frames.

Automatic contour bands during static state display.

During static display of a specific ***state***:
- For each window D3PLOT will find the max and min of all models in that window at that state only.

If you subsequently move on to another state then the contour bounds will change as the new max and min values for that state are used instead.

Commands which display static ***states*** are:
- Any explicit data plotting command (eg **SI**, **CT**, ..) while not animating.
- Using the state slider in the "States" panel.
- Setting an explicit state number or time in the "States" panel.
- Using the <arrow keys> (no shift) to toggle through states.

A useful trick if you want to "animate" a series of states, but to autoscale contours to each state individually, is to use the <arrow keys>: hold them down letting them auto-repeat and D3PLOT will cycle through states with specific contour bands for each state.

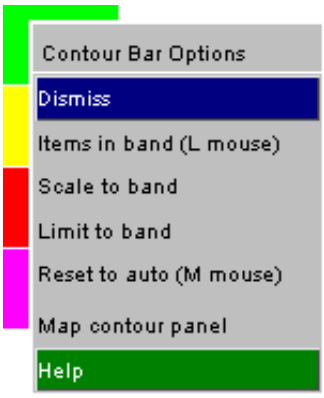
Why is there a distinction between "frames" and "states"?

It is true that for most transient analyses "frames" will be equivalent to "states", however there are some cases where this is not the case:
- When plots are interpolated by time the frame vs state equivalence no longer holds, and typically there are many more frames.
- The user can choose to animate only a subset of the available states, reducing the number of frames.
- In frequency domain (modeshape) analyses each "state" is a mode, and animation frames cycle through +/- 180 degrees at that mode.
- Similarly "static" analyses with a series of loadcases use frames to animate each case in a quasi-modal fashion.

Preserving this distinction makes D3PLOT more flexible and provides more options for contouring animations.

For more details about animation, frames and states, and how to specify them, see section 4.4 on Animation.

4.3.1.2 Clicking on the contour bar to set levels.



As an alternative to the explicit methods of setting contour bands described above it is also possible to set and restrict values by clicking on the contour band display itself.

On a data-bearing plot hover the cursor over the contour level bands, and the cursor symbol will change to `CONT/OPTS`, and the following options are then available:
- Left mouse: limits the display to only those items within the selected band
- Middle mouse: reverts to automatic contour levels
- Right mouse: maps the options menu shown here

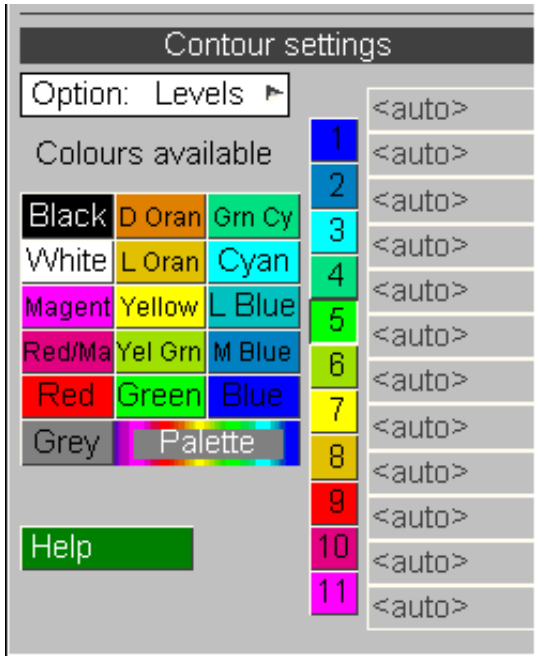
Items in band (left mouse)	Restricts the display to those items in the selected band, but does not alter the overall contour band limits. The effect is based on the centre value of elements, so if contouring is on (the default) you may see gradations of value outside the limits of the band chosen. To prevent this turn averaging off, or select a plotting mode (such as **CL**oud plots) which shows centre values only.
Scale to band	Resets contouring to max/min using the upper and lower values of the band chosen. Display is not limited to this range, so items outside the range will still be drawn.
Limit to band	As **Scale to band**, but also sets limiting values to the original band's max/min, so only elements within the original band are shown.
Reset to auto (middle mouse)	Resets contouring to automatic, and turns off limiting values if switched on by one of the options above.
Map contour panel	Is the equivalent of selecting **Contour > Levels**.

Restriction to a band is carried out using the **Contours > Limiting Values** function described in section 4.3.5. In effect the functions here set the upper and lower bounds, and turn on limiting. You can adjust these further by hand if you wish.

4.3.1.3 Changing contour band colours.

By default colours range from **blue** (low value) to **magenta** (high value), and the colour range is set up automatically. Internally there are 15 standard colours to choose from, and any contour band may be assigned any colour.

To change a colour click on its number (here band 5 has been selected) then select an alternative standard colour from the panel, or use Palette to define your own arbitrary colour.



RESET Will reset contour colours to their default range for this number of levels.

REVERSE Will reverse the current colour range for this number of levels.

Note 1: Contour bands define the upper and lower values of each discrete band. For solid contoured plots (ie **CT**, **SI**) each band lies between these limits. For line contoured **(LC**) plots each line will lie at the mid-point of its band.

Note 2: Colour tables for contours are stored separately for each number of contour levels. So if you change colours for (say) 6 contour levels this will not affect colours for any other number of levels.

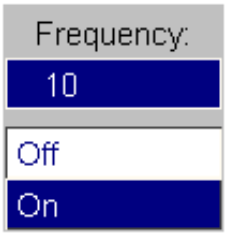
Note 3: During **SI** shaded-image plots the current number of contour levels is mapped onto 21 colour bands, interpolating linearly, regardless of the actual number of bands selected. This is to improve the colour resolution of plots. As a consequence colours are also interpolated within these 21 bands from the **#levels** set here. Thus defining more contour levels will give finer control over the colours used in shaded-image plots.

Note 4: Whichever way they are defined, contour bands must be in ascending, monotonic order. This is particularly significant for **USER_DEFINED** mode: you will not be permitted to create bands that have zero or negative intervals.

4.3.2 LABELS... Setting the labelling of line-contour plots.

By default **LI** line-contour plots on colour displays do not have the contour bands labelled, but it is possible to turn this on, and to control labelling frequency.

This figure (right) shows the **LABELS...** control panel. Labels may be toggled on and off, and the labelling frequency can be set.



Frequency must be an integer value between 1 and 1000 and means "label every nth line". So lower values give more frequent labelling, and higher values sparser labelling. Its default value is 10.

Note: Labelling is not necessary on colour devices, since line colour is sufficient to distinguish between contour levels, however on monochrome and greyscale devices it is needed to tell lines apart. Therefore if you are running on a monochrome display, or on any display with the laser switch on and the laser output set to be in **GREYSCALE** mode, line labelling will be applied regardless of the switch setting above.

Click here for the next section

4.3.3 **RESOLUTION...** Setting contour resolution

The output from 2D & 3D elements in LS-DYNA is only written at element centres, implying constant stress: no values are written at nodes on elements.

Therefore contouring, by displaying variations of data across elements, is an approximation used to help visualisation of the data distribution.

Three levels of contour resolution are provided:

LOW Data are averaged at nodes, then contour bands (or lines) are drawn linearly from edge to edge across elements. The element centre values are not directly included so some smoothing of results occurs and peak values may be lost: which is potentially unsafe.

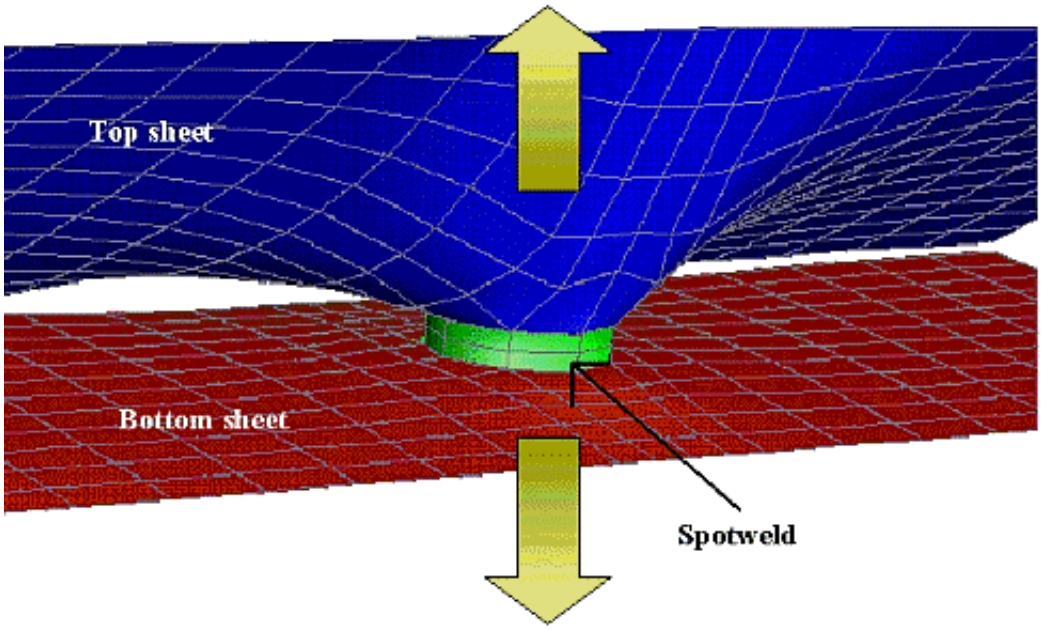
MEDIUM The value with the greatest magnitude at each node is found, then contour bands (or lines) are drawn linearly from edge to edge across elements. The element centre values are implicitly included since they will qualify as maxima at nodes: a safe overestimate.

HIGH Data are averaged at nodes as before, but elements are then split into sub-areas using centre and mid-side values. This enables variations across elements to be seen, and peak centre values included. However it requires up to eight times as much computation, graphics storage and drawing effort as the other two modes.

The default mode is **MED**(ium) resolution since this is both "safe", (peak centre values are included, albeit smeared out to element vertices), and quick (computation, graphics data storage and drawing effort are small).

To understand the effects of the three possible contour settings consider the following example:

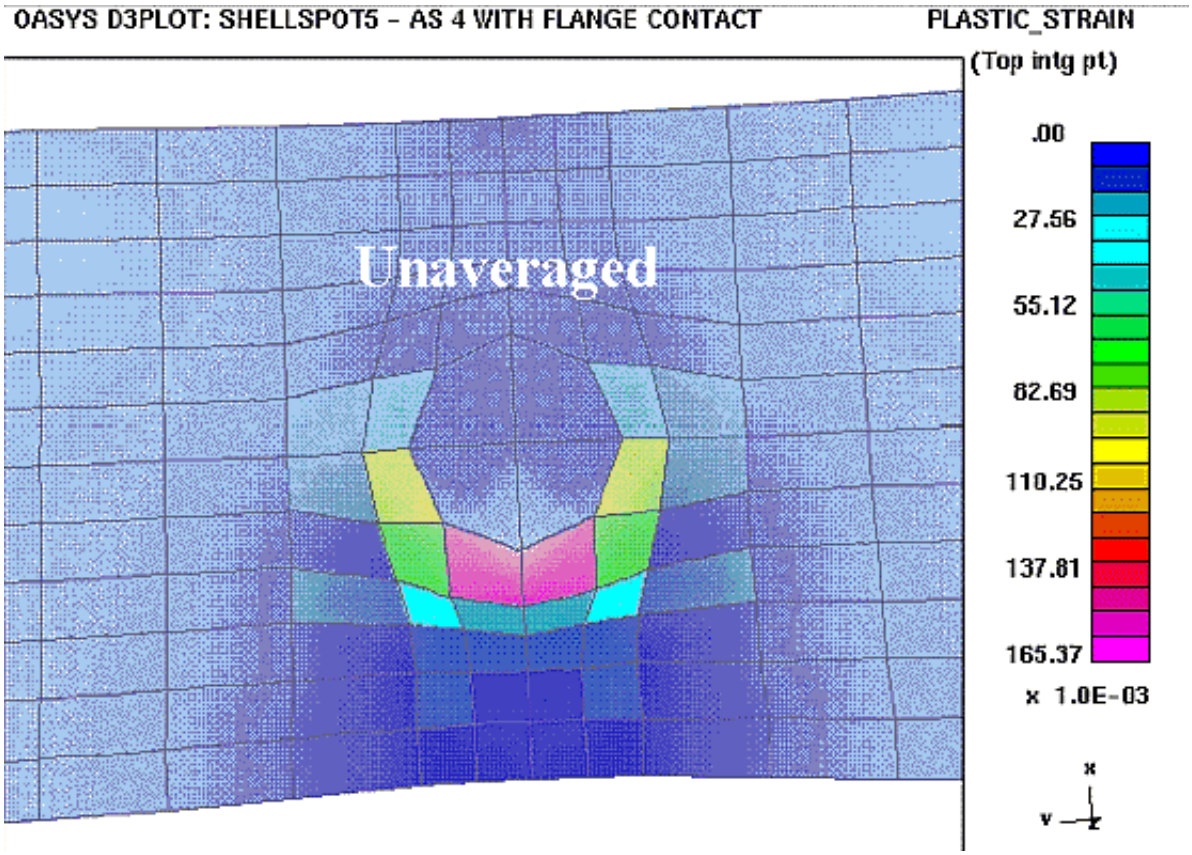
Two sheets are spot-welded together, and are then pulled apart.



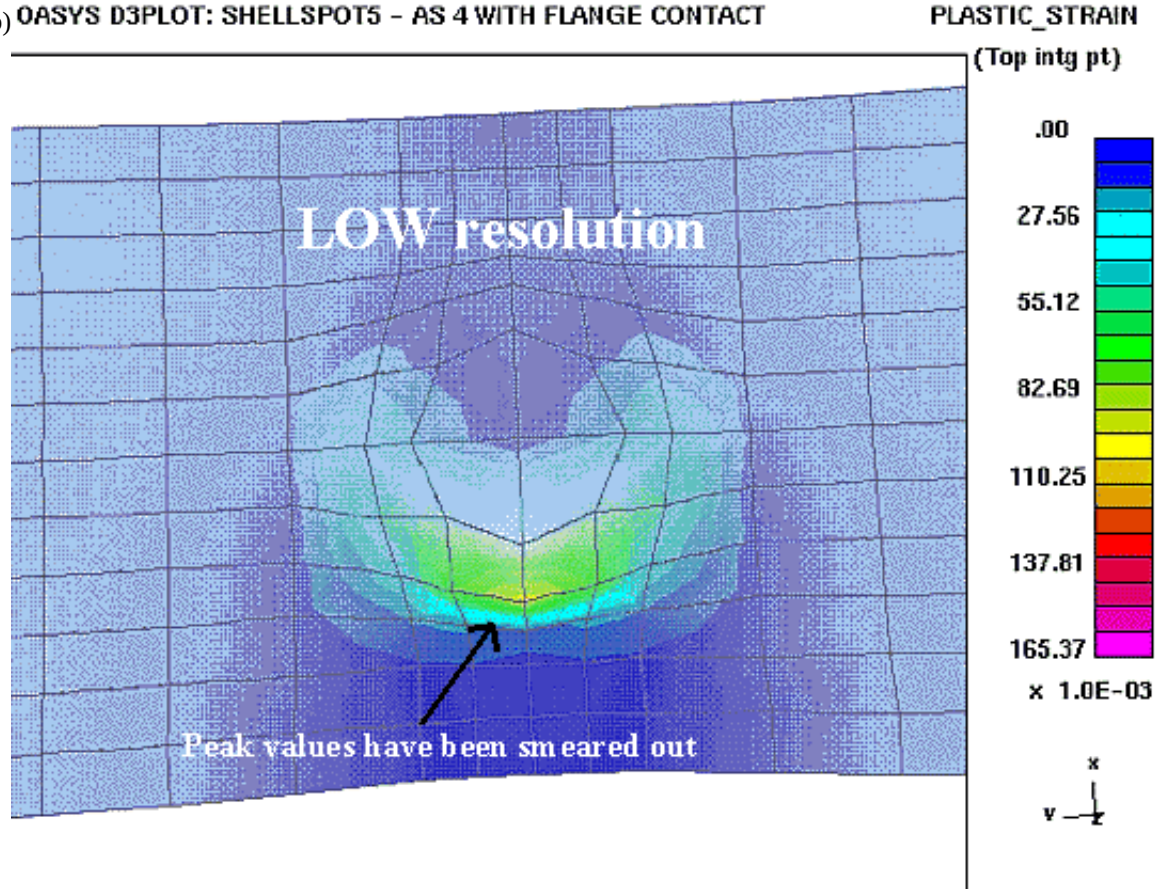
The following plots (a - d) show the plastic bending strain in the top sheet as:

- **Un**averaged
- **LOW** resolution
- **MED**IUM res
- **HIGH** res

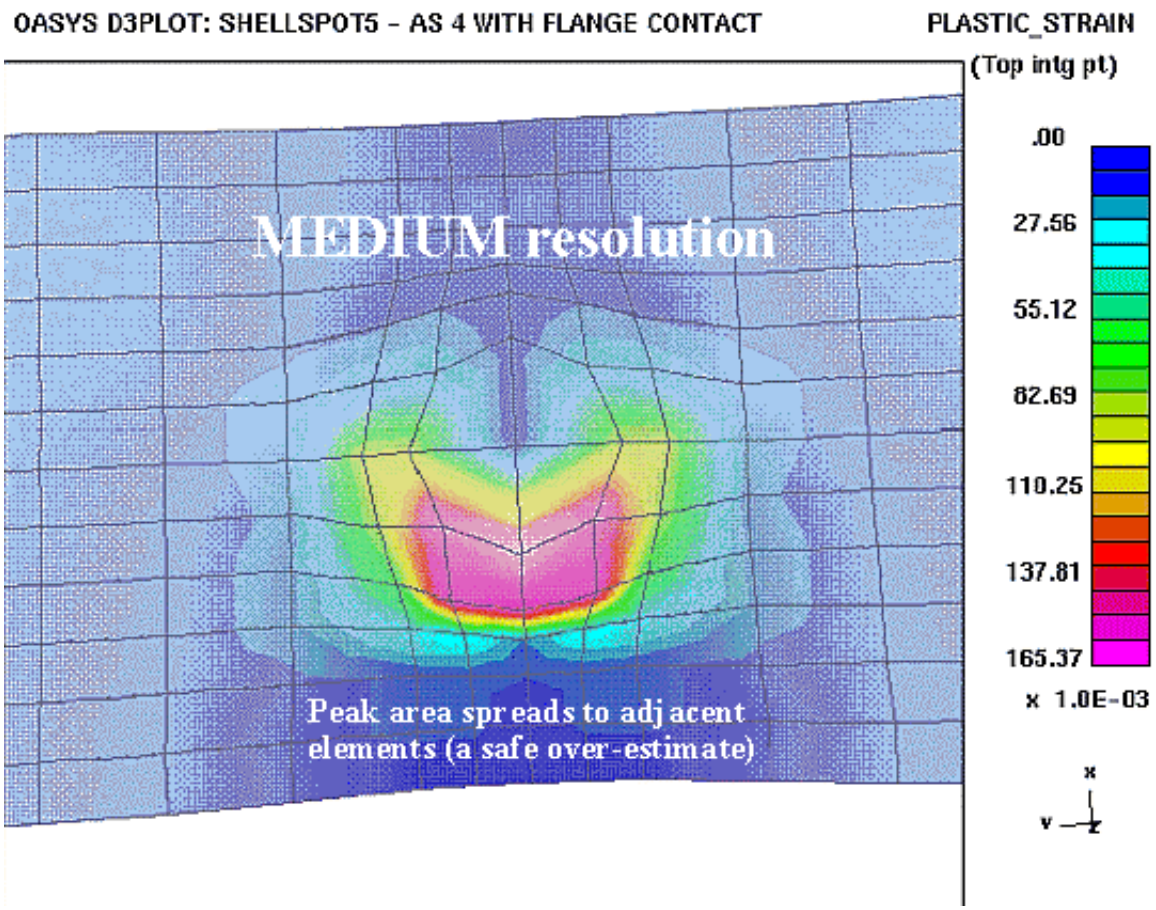
(a)


OASYS D3PLOT: SHELLSPOT5 - AS 4 WITH FLANGE CONTACT
PLASTIC_STRAIN
(Top intg pt)
.00
27.56
55.12
82.69
110.25
137.81
165.37
x 1.0E-03
Unaveraged

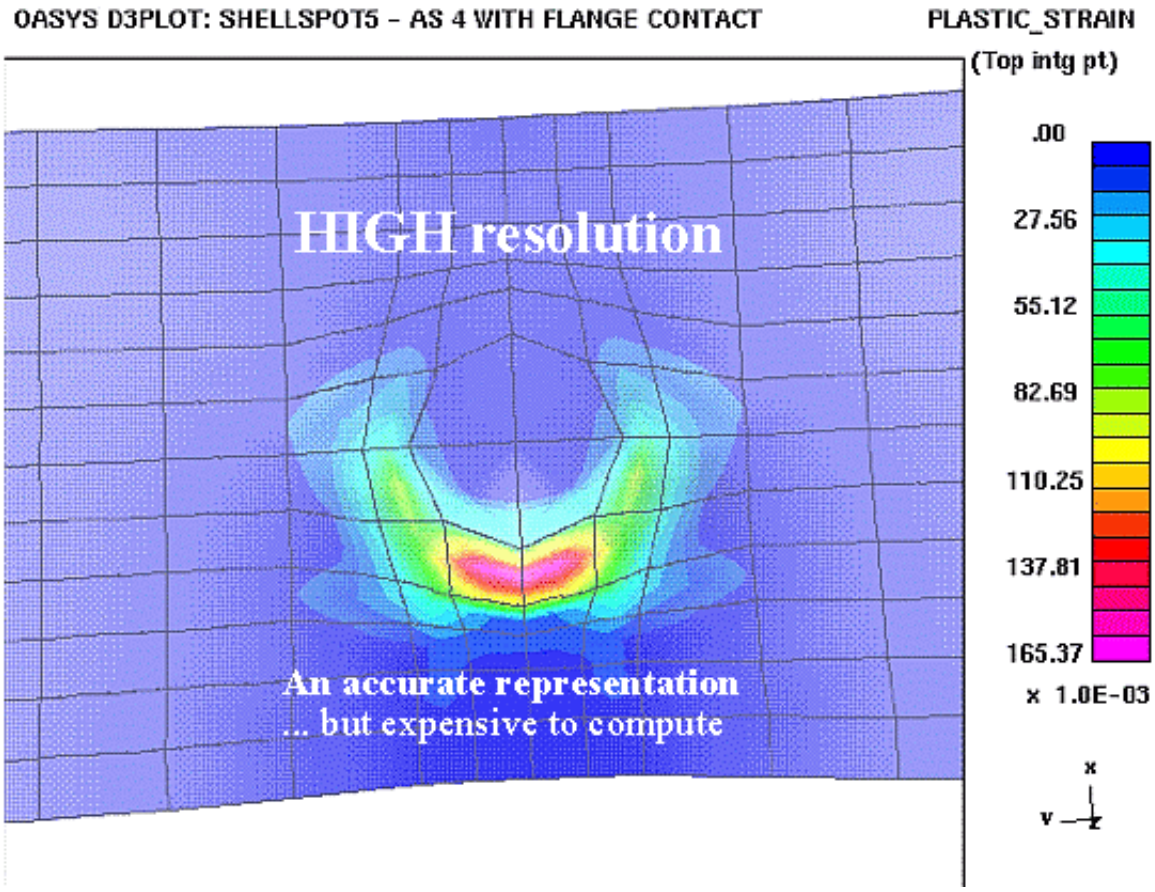
(b)


OASYS D3PLOT: SHELLSPOT5 - AS 4 WITH FLANGE CONTACT
PLASTIC_STRAIN
(Top intg pt)
.00
27.56
55.12
82.69
110.25
137.81
165.37
x 1.0E-03
LOW resolution
Peak values have been smeared out

(c)


OASYS D3PLOT: SHELLSPOT5 - AS 4 WITH FLANGE CONTACT
PLASTIC_STRAIN
(Top intg pt)
.00
27.56
55.12
82.69
110.25
137.81
165.37
x 1.0E-03
MEDIUM resolution
Peak area spreads to adjacent elements (a safe over-estimate)

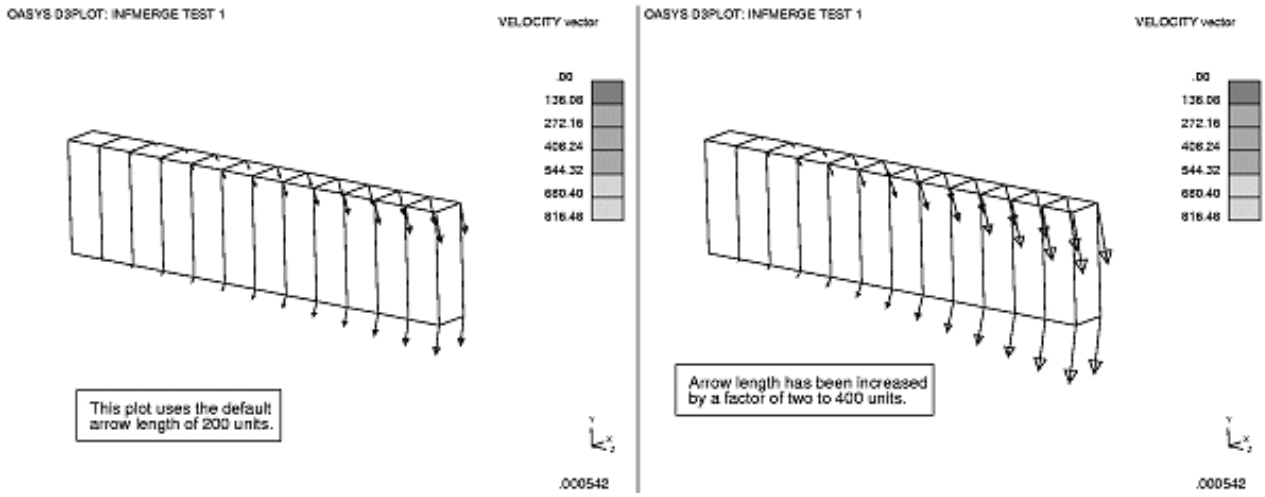
(d)


OASYS D3PLOT: SHELLSPOT5 - AS 4 WITH FLANGE CONTACT
PLASTIC_STRAIN
(Top intg pt)
.00
27.56
55.12
82.69
110.25
137.81
165.37
x 1.0E-03
HIGH resolution
An accurate representation ... but expensive to compute

4.3.4 ARROWS... Setting arrow length in vector plots.

VEL velocity plots, and **VEC** vector plots of contact force, both use arrows to show the direction and magnitude of the data.

Arrow length is set by default such that the maximum contour band value gives an arrow 200 screen units long, other values being scaled in proportion. The figures below show examples of a velocity plot with default and double length arrows.



This figure shows the **ARROWS** control panel.

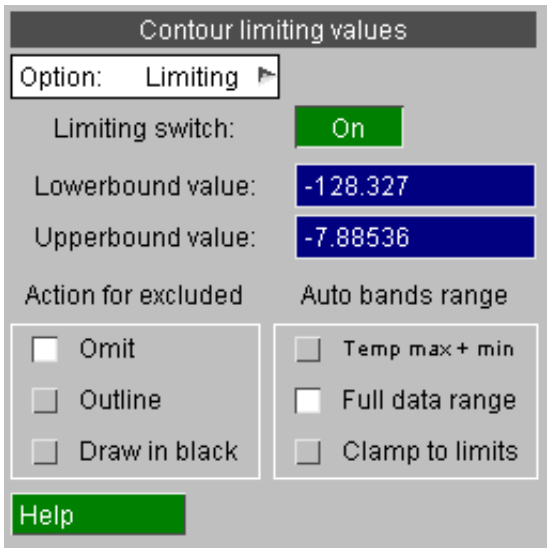
Length is entered in screen units, here 400 has been used.

4.3.5 LIMITING_VALUES... Limiting what is contoured by value range

By default all elements are contoured, regardless of whether or not they lie within the max & min range of the contour bands specified. Elements lying outside these bands are given the highest or lowest contour band colour as appropriate.

In some cases you may wish to contour only those elements which lie within a restricted range of values, and to omit (or draw only in outline) those outside this range. This can be done with the **LIMITING_VALUES** option.

This figure shows the control panel for the **LIMITING_VALUES** options.



When the **Limiting switch** is off, (default), the other options in this panel are greyed out.

Lowerbound value	is the value below which data are not contoured.
Upperbound value	is the value above which data are not contoured.
Action for excluded	defines what happens to excluded elements, see below.

You need to consider what action is to be taken with elements that are not contoured. There are three options, which it is convenient to take in reverse order.

The **`Auto bands range`** options determine how Limiting Values interact with automatic contour bands when limiting is active:

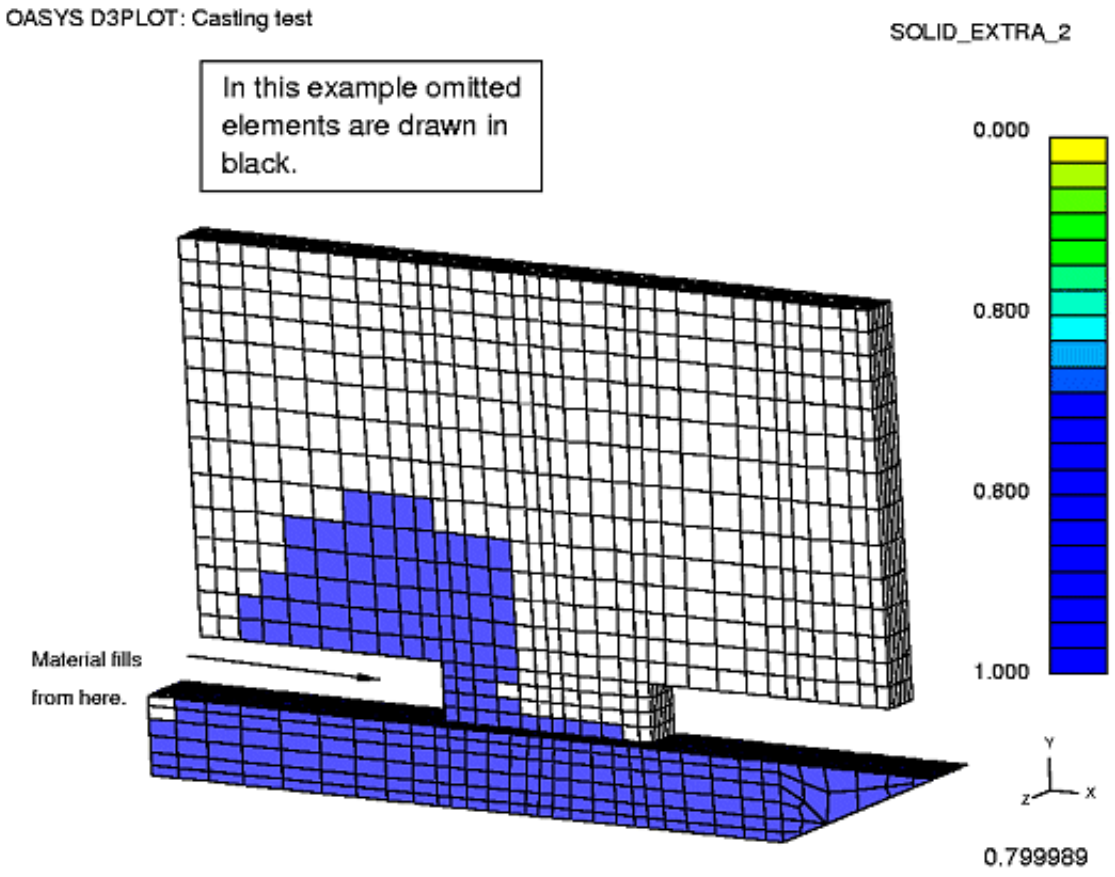
`Temp max + min`	Is used when the short cut "click on a contour band" option in section 4.3.1.2 has been used. This temporarily sets contouring to enforced max/min values, which conflicts with this mode, and should not be set manually.
`Full data range`	If automatic contour bands are in use then they will adjust themselves to the full range of data available, ignoring the lower and upperbound values set here. Therefore the contour band boundaries will not change as the upper and lowerbound values are changed here.
`Clamp to limits`	Restricts automatic contour bands to the limits set here, changing as they change. Therefore whatever is drawn on the plot will exploit the full range of contour bands and colours available, making this suitable if you want the maximum colour and band discrimination for what is drawn.

The following three examples are a mould filling (fluids) analysis in which a plate comprised of solid elements is filled from below. They show the effects of the three **Action for excluded** options, in particular note how this affects the display of internal structure:

This figure uses **DRAW_IN_BLACK** for elements that lie outside the limiting values. (Black is reproduced as white on a hardcopy as here.)

Here we are plotting "void fraction", ie %age fullness of fluid, and only values in the range 0.8 to 1.0 (ie 80% full or more) are shown.

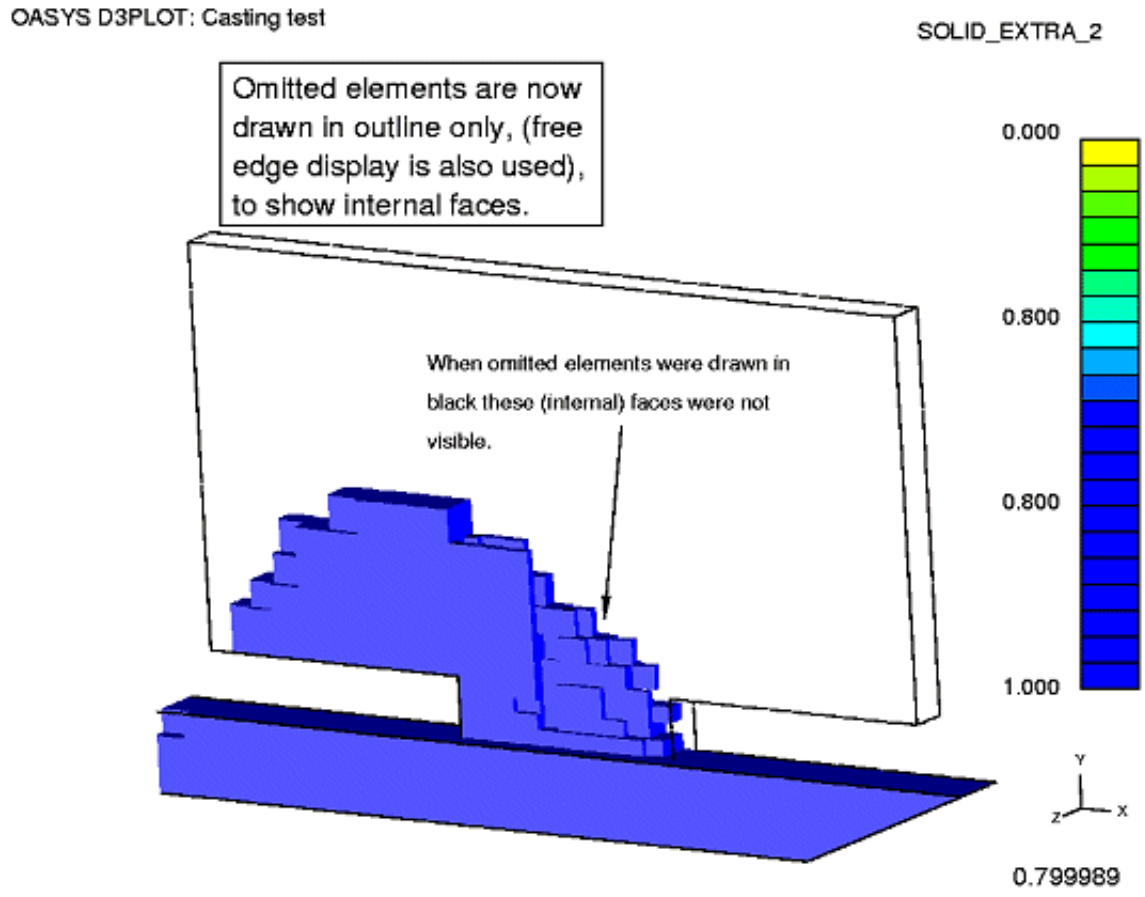
It is clear that no internal structure is visible since elements on the outside faces, which are not full, obscure it.



This figure uses **OUTLINE** for elements that lie outside the limiting values.

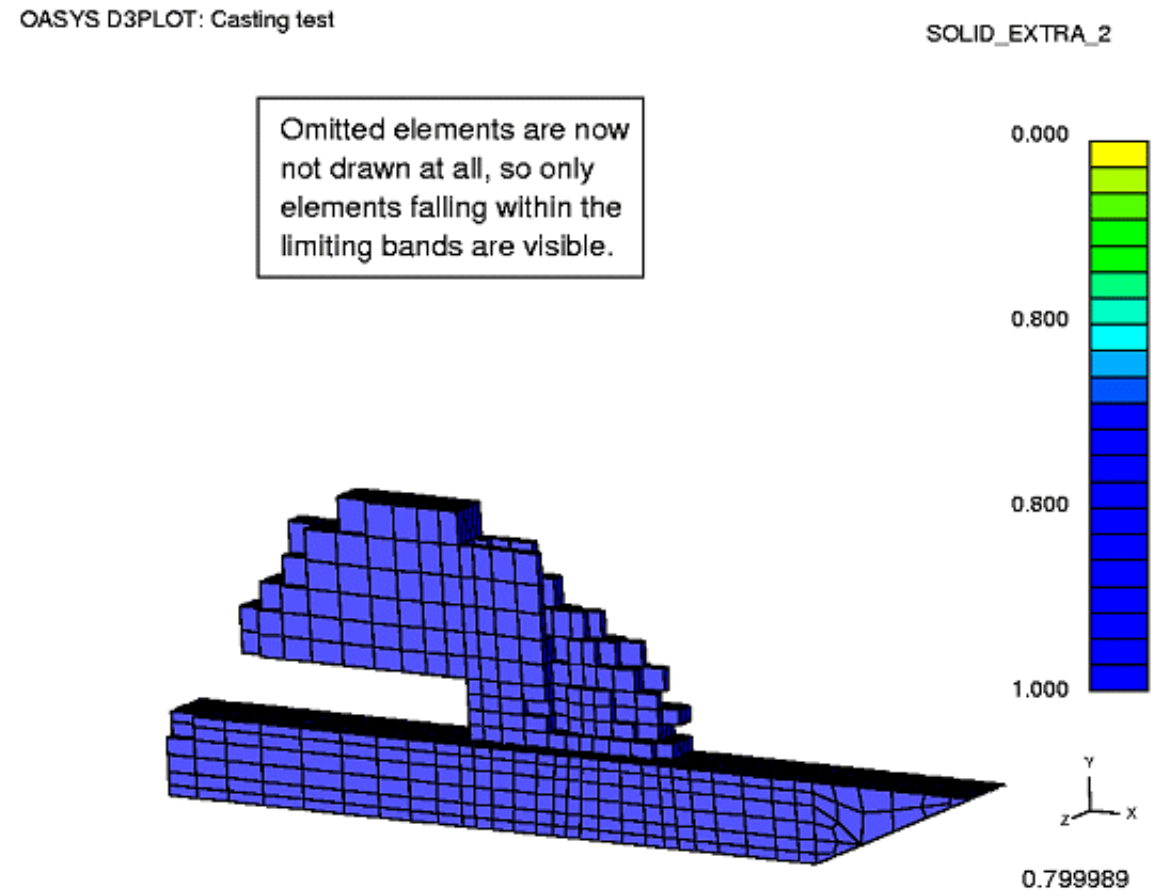
Only the outline of elements which are out of range is drawn, so that they do not obscure structure behind them.

In this example free edge display (**[DISPLAY_ OPTIONS] FREE_EDGE**) has also been used to remove clutter of excess mesh.



This figure uses **OMIT** for elements that lie outside the limiting values.

Now only those elements that are within the specified limits are drawn at all, those outside them are effectively blanked from the display.



The default state of the **LIMITING_VALUES** switch is off, so that none of these settings apply.

When turned on it will apply to any data plotting mode, but clearly the **OUTLINE** and **DRAW_ IN_BLACK** options for excluded elements are only meaningful for 2D and 3D elements.

4.3.6 **DITHERING...** Dithering of contours in **SI** shaded-image plots

This is a simple switch that toggles the dithering of contours in **SI** shaded-image plots only, it has no affect on other plotting modes. Its default state is OFF, ie no dithering.

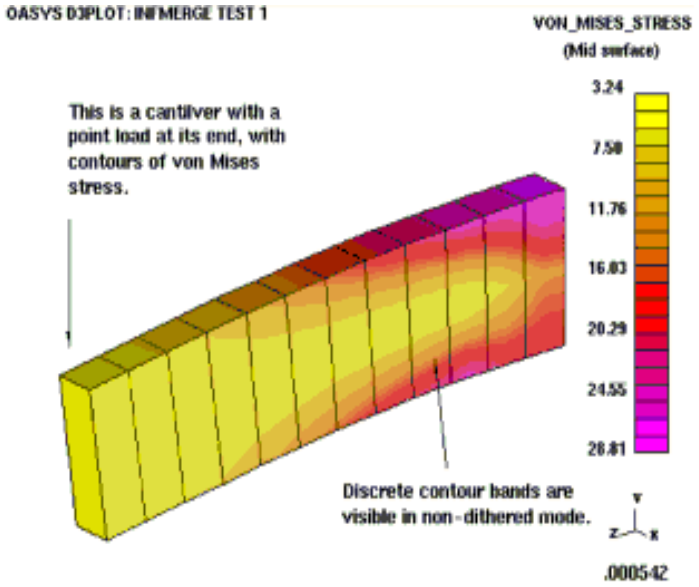
Note: Users on 3D devices will do better with **GOURAUD** contours option for SI plots, which is the default: see Section 4.3.7. and should ignore this option which is only relevant to legacy 2D graphics.

This figure shows an un-dithered shaded-image plot.

Discrete contour bands are visible since they are drawn as polygons.

This is much quicker to compute and draw than the dithered image below, and laser files are smaller, but the image quality is not so good.

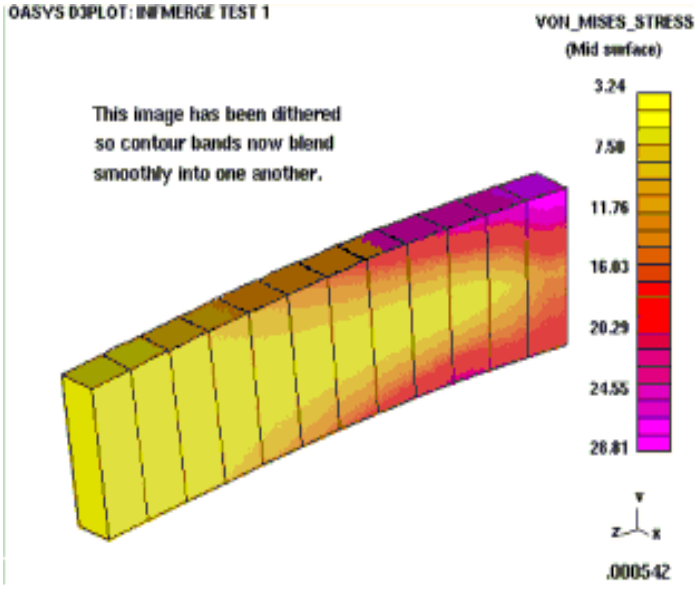
This is the default mode since it is quick.



This figure shows the same model, but with dithering turned on.

There are now no discrete contour bands, they have been blended smoothly into one another by the dithering process.

But this image takes much longer to compute and draw, and any laser file will be much larger because a value is written at every pixel.



4.3.7 **GOURAUD...** On 3D devices only: Gouraud shading of contours on SI shaded-image plots.

This is a simple on/off toggle switch for the gouraud shading of contours on shaded-image plots.
It is on by default if the hardware support is available.

Gouraud shading applies:
- Only on 3D devices, operating in 3D mode, which have hardware support for it;
- Only to the contours of **SI** shaded-image plots in this situation.

It achieves a result similar to the dithered image in the dithered shaded-image plot, but does this using hardware interpolation of shades rather than software generated dithering. If your hardware supports it you should use this in preference to dithering: it will give better looking results, and will run about 100 times faster. For this reason it is the default setting for shaded-image plots on 3D devices.

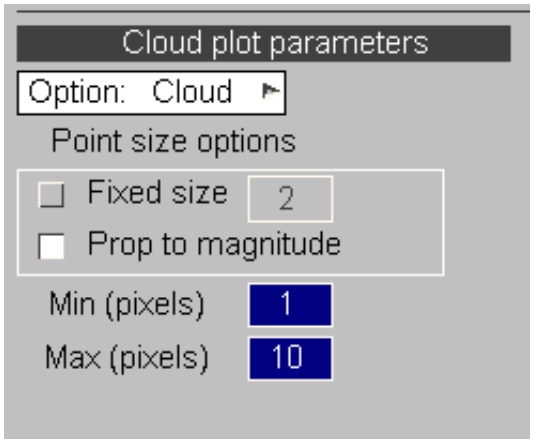
Note 1: Hardware generated gouraud shading cannot be sent to laser plots, since laser images have to be generated in software. Turning the laser on will cause the display to revert to 2D mode giving, by default, contour bands as shown in the un-dithered shaded-image plot. To send a smoothly shaded **SI** image to a laser file you will have to use dithering as described in Section 4.3.6.

Note 2: Gouraud shaded **SI** images will often be generated faster than apparently simpler **CT** continuous-tone images on 3D devices. This is because fewer polygons are generated (gouraud shading only requires facet vertex, ie nodal, values). In addition 3D hardware is usually optimised to render smooth shaded and lit polygons quickly.

Note 3: Gouraud shaded images are also more economical of memory space than display modes that contain discrete sub-polygons of contour bands in elements. Again, this is because only nodal data values need be sent to the hardware (assuming low resolution contouring). For this reason gouraud shaded **SI** mode is often a good choice for building animations. If speed and memory space are at a premium the recommended settings would be:

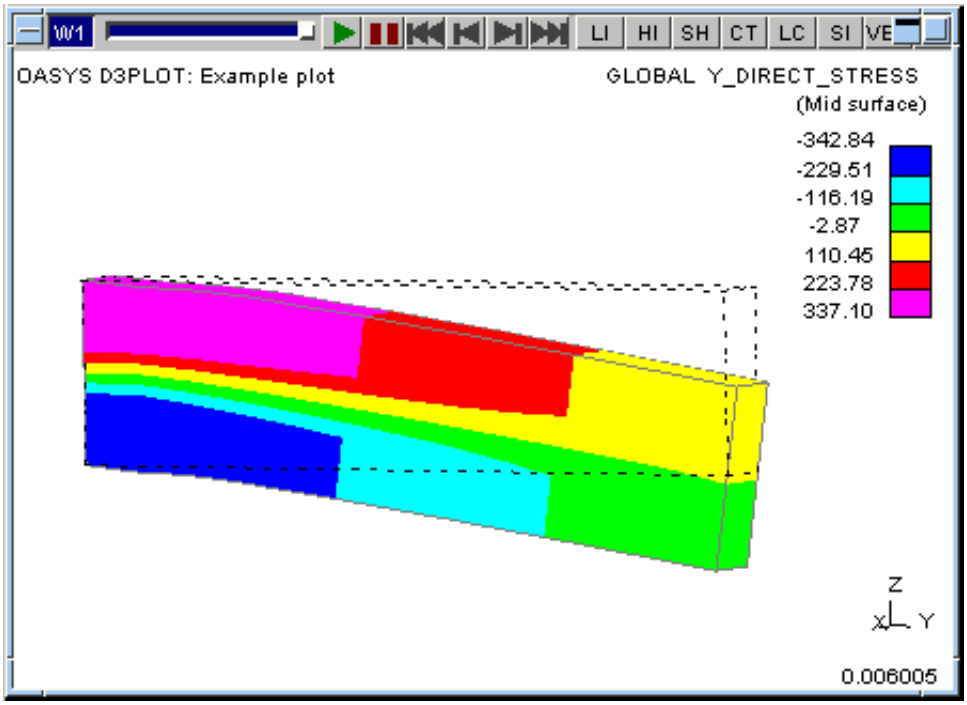
- Gouraud shading **on**;
- **Low** resolution contouring;
- Either overlay **off**; or **on**, but with free-edge display also turn

4.3.8: **Cloud Plots**: Controlling the attributes of **CL** mode plots.

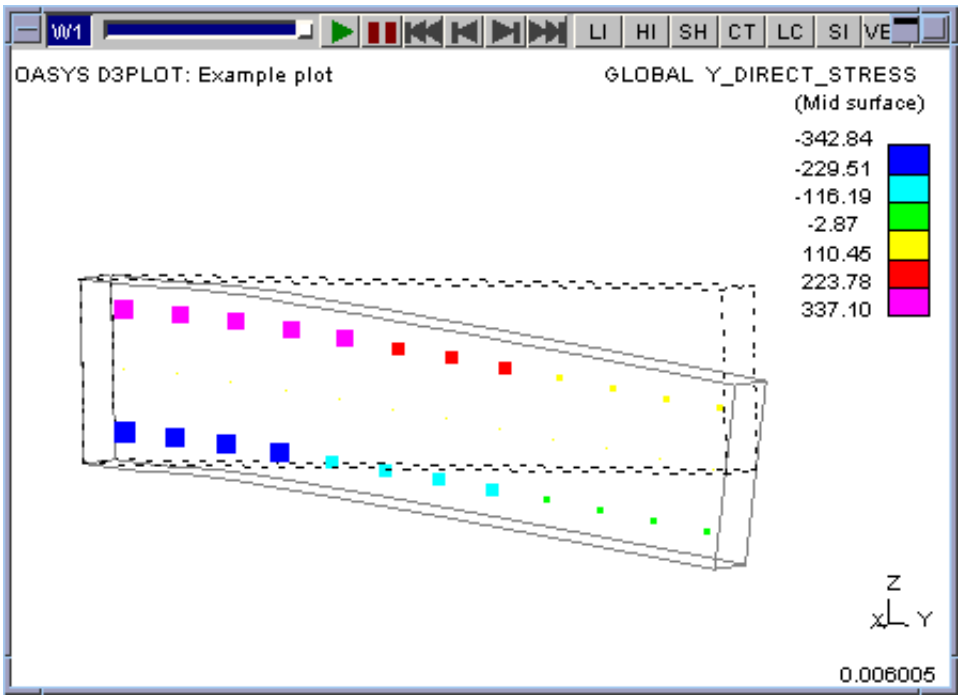


Point size may be:
- Fixed, the default being 2 units
- Proportional to data magnitude, autoscaled to the current contour range.

Here is a simple cantilever of solid elements, loaded in bending, and plotted in CT mode.



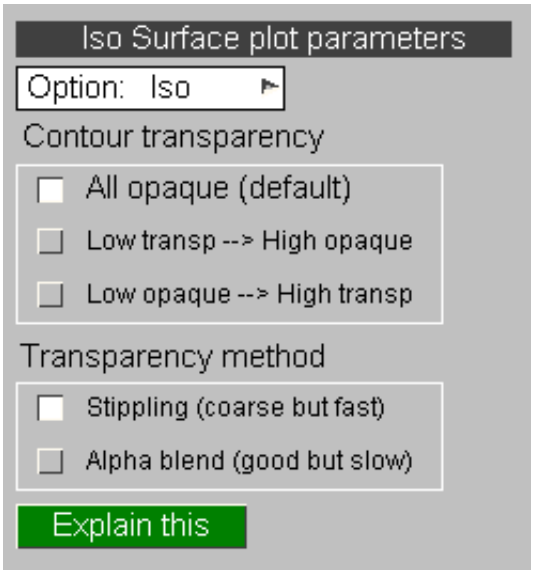
And here is the same model, this time as a Cloud plot with variable symbol size:



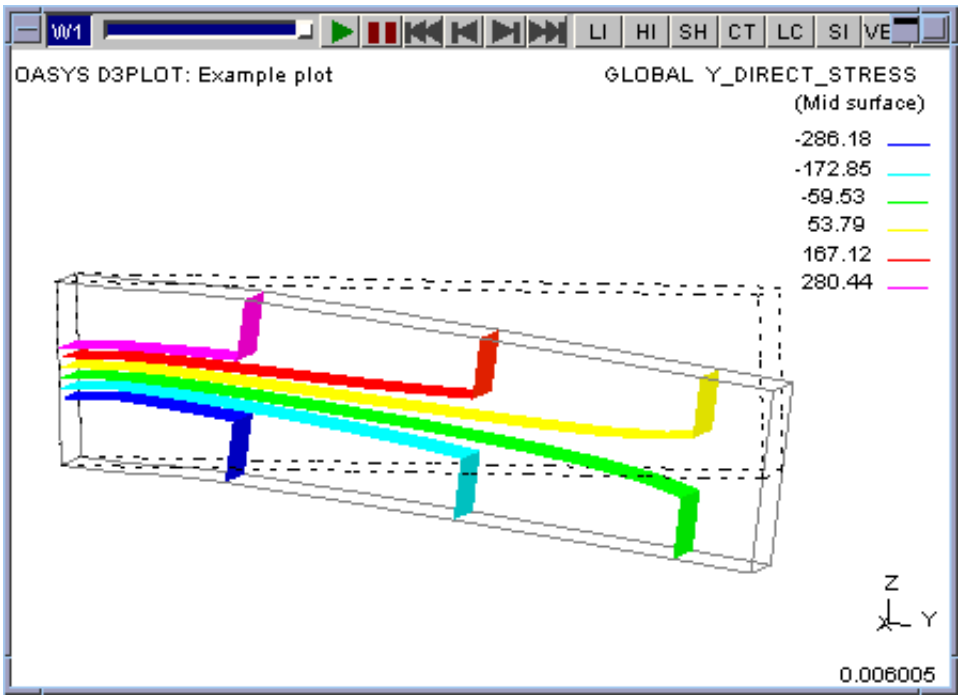
4.3.9: **Iso Plots**: Controlling the attributes of **ISO** mode plots.

By default ISO plots are opaque, but this can osbcure internal detail so it is possible to make them transparent.

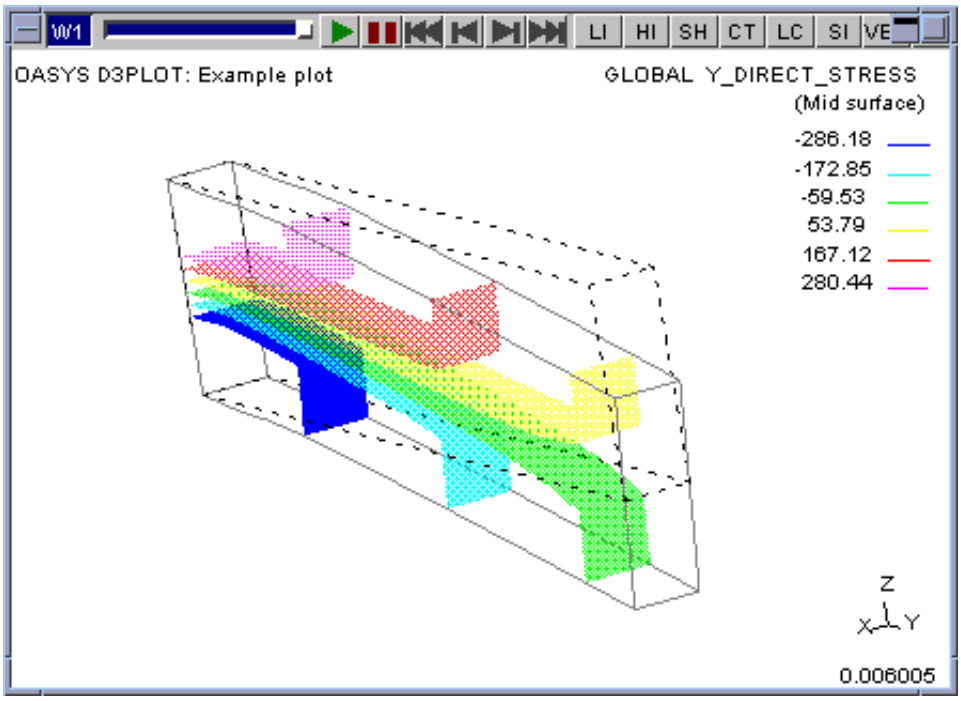
Transparency on graphics hardware is not perfect and, moreover, can be slow to render. So two alternative methods of displaying transparent ISO contours are provided.



Here is the same cantilever as above, rendered as a default (opaque) ISO plot:



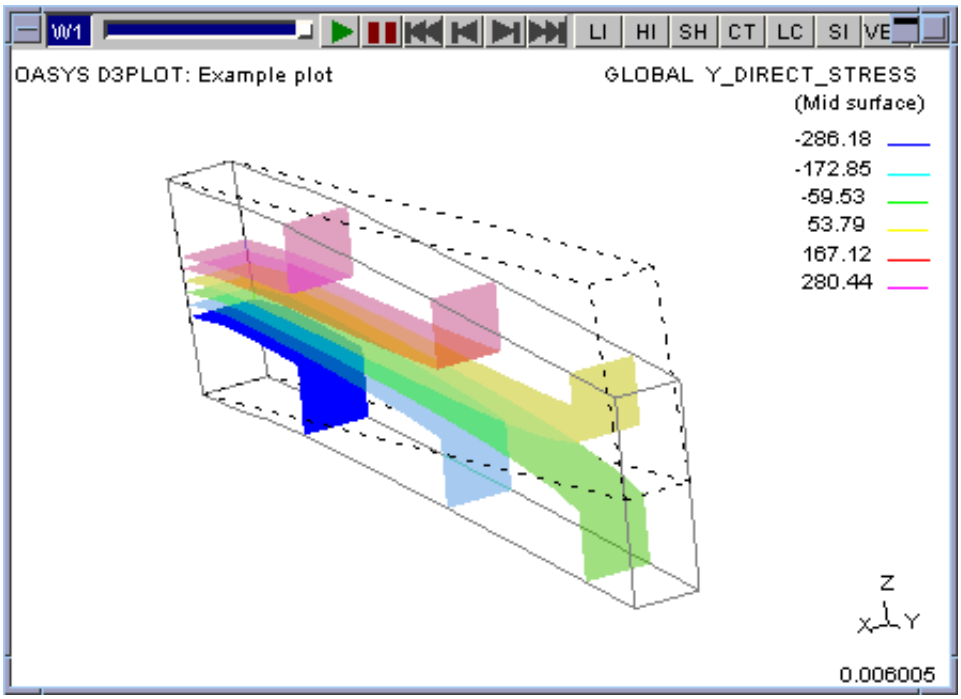
This is the same model using transparent "stippled" contours, with the lowest values (blue) rendered as opaque.



And here is the same image again, rendered using "Alpha blending".

The quality of the transparency is much better although it would be slower to draw with a large model.

However ...

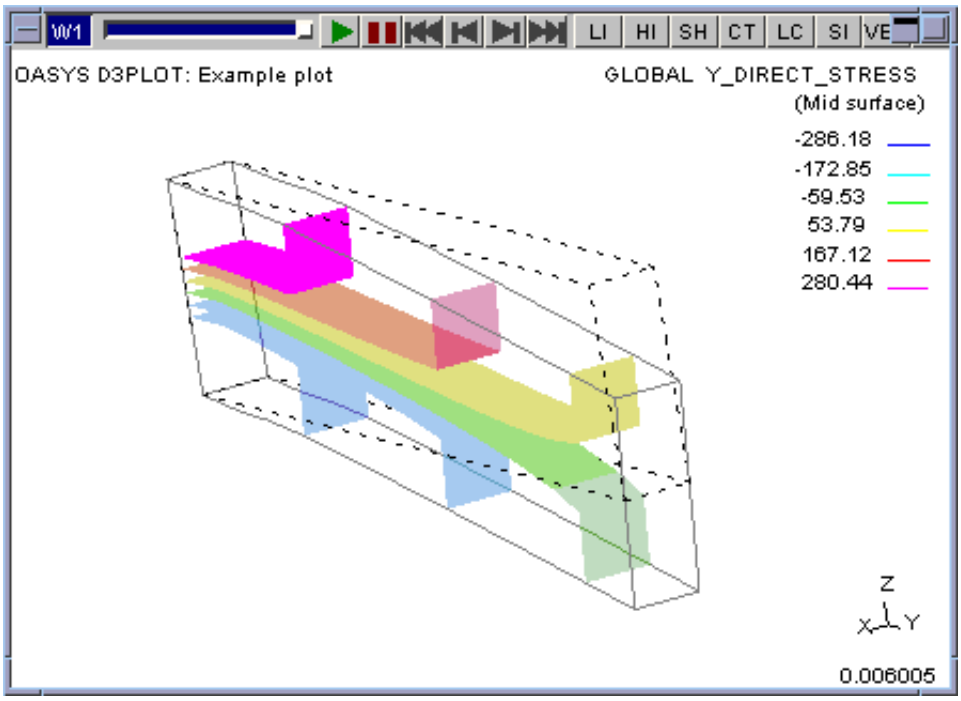


... using Alpha transparency does not always give the "right" answers.

Here is the same image, this time with the high values opaque, and it is clear that the lower colours are not visible when they are behind the higher ones.

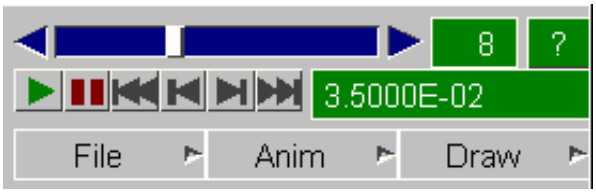
This is because "Alpha blending" in the hardware is not, in itself, always a totally reliable way of producing realistic transparency because the results depend on the order in which facets are drawn.

It may be necessary to revert to stippling, which will always give "correct" precedence of facets - albeit with a poorer spatial resolution - to get acceptable results.



Click here for the next section

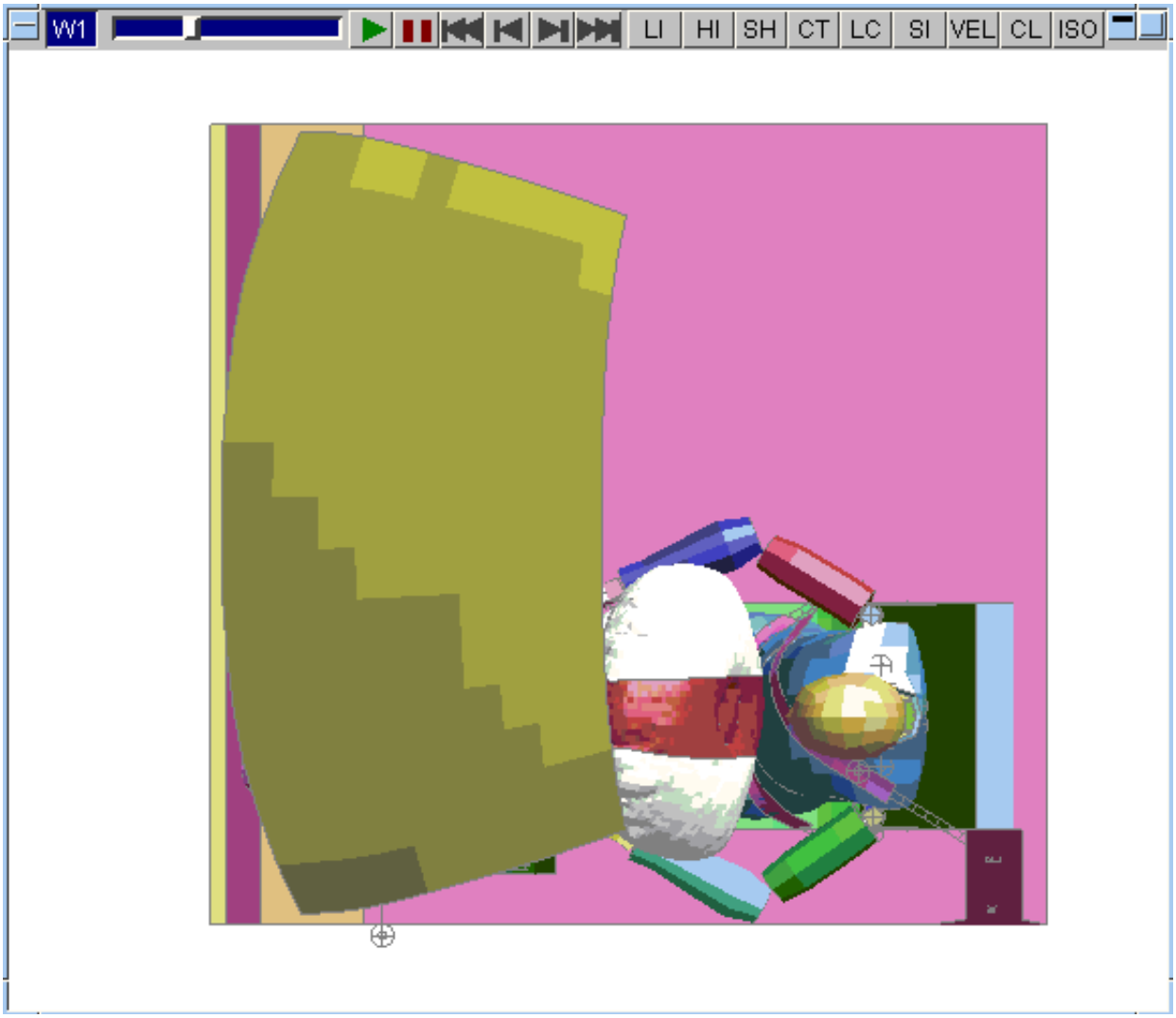
4.4 Animation **How to display, control, store and retrieve animation sequences.**



In D3PLOT 8.0 there is no distinction between "static" and "animation" modes: a static plot can be thought of as a frozen slice of a dynamic animation sequence. Any conventional display mode can be animated at any time by pressing **PLAY >**, and halted again with **STOP**.

Virtually all menu functions such as blanking, component change, and so on can be used while animating, as can dynamic viewing: it is not necessary to halt an animation in order to change the attributes of your image.

4.4.1 Basic animation controls.



As well as the **PLAY >** and **STOP** controls in the **State Display** box the graphics window has a set of controls at its top left which can be used to control animation, and the state used for static display.

The master slider in the **State Display** panel controls all graphics windows for which its **W*n*** tabs are active.

The slider and associated controls at the top of a graphics window control that window only.

STOPs the current animation.

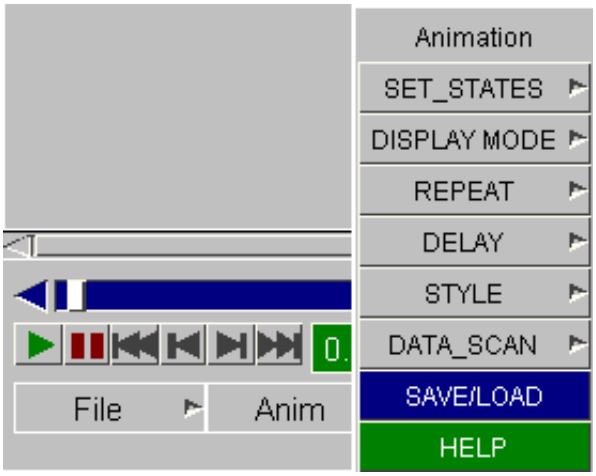
Starts animation (same as **PLAY >**)

Jump straight to first or last frame
Single step backwards (**|<**) and forwards (**>|**)

The state slider allows you to scroll with the mouse to any state, and to slide dynamically through them.

If an animation is playing using any of these controls (other than **PLAY**) will automatically stop it in order to execute the new commands.

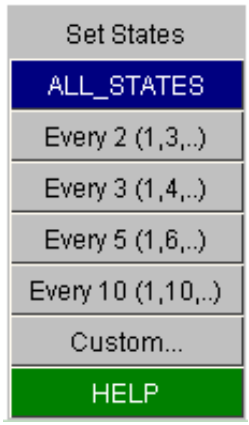
4.4.2 **ANIM >** Controlling the animation process.



The **ANIM >** popup menu gives access to the controls for animation.

Each option has a set of standard sub-menus, and further "custom" defaults for more complex settings.

4.4.2.1 **SET_STATES >** Selecting the states to be animated



By default all states are selected for animation. You can select from the pre-programmed **Every n** options here, or use **Custom...** to define more precisely what is to be animated.

The **ALL_STATES** and **Every *nn*** options will apply to all windows with active **W*n*** tabs in this panel.

Where multiple models are present the **Custom...** states can only be selected for one model at a time. You should set the **W*n*** tabs to the window(s) of this model first: attempting to select custom states for multiple models will generate warnings.

Interpolating animations by time

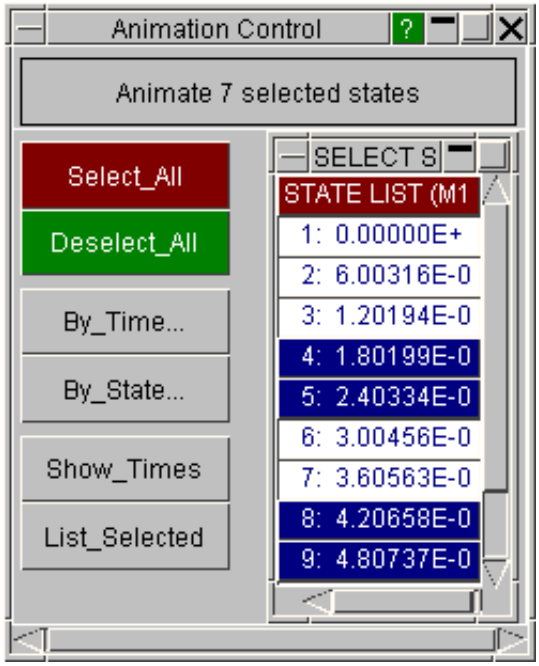
Normally animations are drawn at the states defined in your database(s), but it is possible to interpolate by time between states: both to get a smoother progression through a sequence with too few states, and also to match multiple models with dissimilar output frequencies. This is described under "custom" animation **BY_TIME...** below.

Animating Static and Eigenvalue (modeshape) analyses.

This section assumes the normal case of a transient analysis producing a series of states at successive times. However it is also possible to animate a single static or eigenvalue state by cycling it through 0 - 360 degrees. This is described in section 4.4.5.

Custom animation definition

You can select any permutation of states from the **STATE LIST** menu. (In this example states 7, 8, 11, 12 have been de-selected.)



(DE-)SELECT_ALL	(De-)selects all states in the **STATE LIST** menu.
SHOW_TIMES	Lists all available states in the database file.
LIST_SELECTED	Lists more details about the currently selected states.
BY_STATE...	Allows you to select states via <start> <increment> <end> syntax. (Convenient for models with very many states.)
BY_TIME ...	Interpolating between states by defining time intervals.

Note that this "custom" panel can only apply to one model at a time. When multiple models are present it will be restricted to the window(s) of a single model.

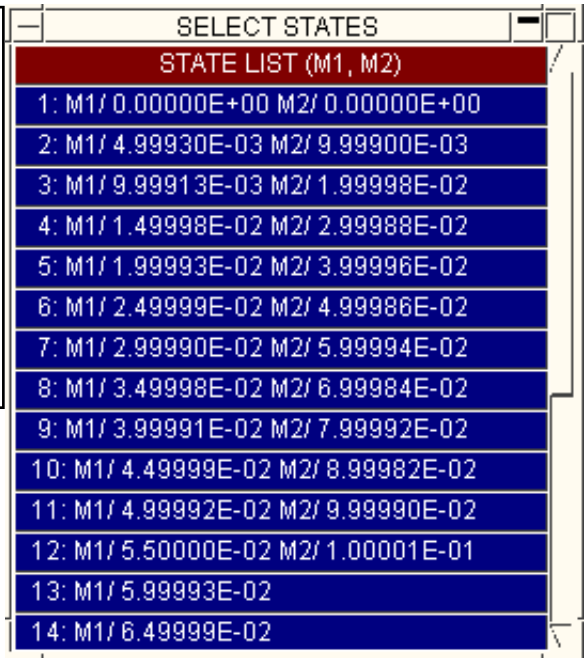
How models with dissimilar states are animated.

The examples above consider the case of only one model. Where two or more models exist it is possible that they will have a different number of states, at different time intervals.

Consider the following example of two models with dissimilar states:

Model 1 states	**Model 2 states**
1. T = 0.000	1. T = 0.000
2. T = 0.100	2. T = 0.200
3. T = 0.200	3. T = 0.400
4. T = 0.300	4. T = 0.600
5. T = 0.400	5. T = 0.800
6. T = 0.500	6. T = 1.000
7. T = 0.600	
8. T = 0.700	
9. T = 0.800	
10. T = 0.900	
11. T = 1.000	

The **SELECT STATES** menu in this example will now show how the states align by frame, as follows:



Animation sequence when models have dissimilar numbers of states

In this situation a model with fewer states waits until the one with more states has finished its sequence before looping back to zero. Using the example models above we now get:

Frame	Model 1 shows	Model 2 shows
1.	• T = 0.000	• T = 0.000
2.	• T = 0.100	• T = 0.200
3.	• T = 0.300	• T = 0.400
4.	• T = 0.400	• T = 0.600
5.	• T = 0.500	• T = 0.800
6.	• T = 0.600	• T = 1.000
7.	• T = 0.700	• T = 1.000 < = Hold last state
8.	• T = 0.800	• T = 1.000 < = Hold last state
9.	• T = 0.900	• T = 1.000 < = Hold last state
10.	• T = 1.000	• T = 1.000 < = Hold last state
11.	• T = 0.000 **<= Loop back to state 1**	• T = 0.000 **< = Loop back to state 1**
12.	• T = 0.100	• T = 0.200
13.	• T = 0.200	• T = 0.400

Note that the "holding" operation in M2 is based on state number, not time.

This "hold last state" logic applies whether the dissimilar models are in the same or different windows. (This is a change of behaviour in D3PLOT V92, in previous versions there was no synchronisation between windows, and models were only "held" if they were in the same window.)

What the "clock" in a graphics window shows with two or more models in a window.

Where there is only one model in a window then there is no ambiguity, and the clock at the bottom right shows the current state's time.

Where there are two or more models in a window then the clock shows:

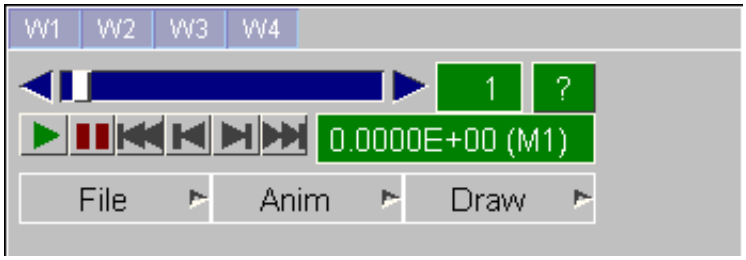
- **`max (Time of M1, Time of M2, ...)`**
- If the times in the multiple models do not match within 0.1% then the clock's colour is inverted, typically black on white.

Synchronising animations by state across multiple windows.

From release 9.2 onwards animation in multiple windows is aligned by state number. Thus all animations will start at state #1, and step forwards together through states #2, #3, .. #n. If a window has fewer states than one or more other windows it will wait at its last state until the window with the highest number of states has reached its end, then they will all loop back to state #1 together.

The **STATE NUMBER** slider will set the selected state for all active windows (here **`W1 .. W4`**), stopping animation if it is currently running.

If you subsequently restart animation with the master **PLAY >** button in this panel then all windows will commence from the same state and, if they have the same number of frames, will remain synchronised.

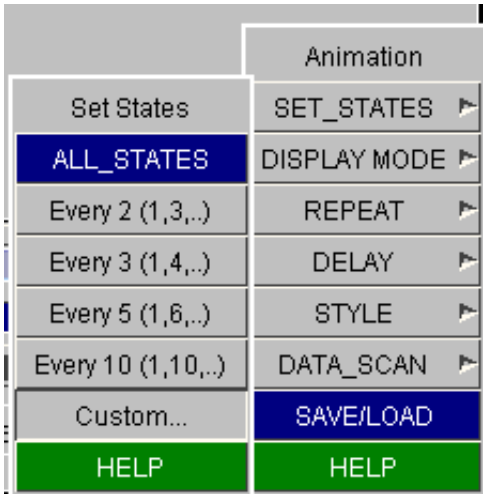


Synchronising animations by state when windows have different numbers of states.

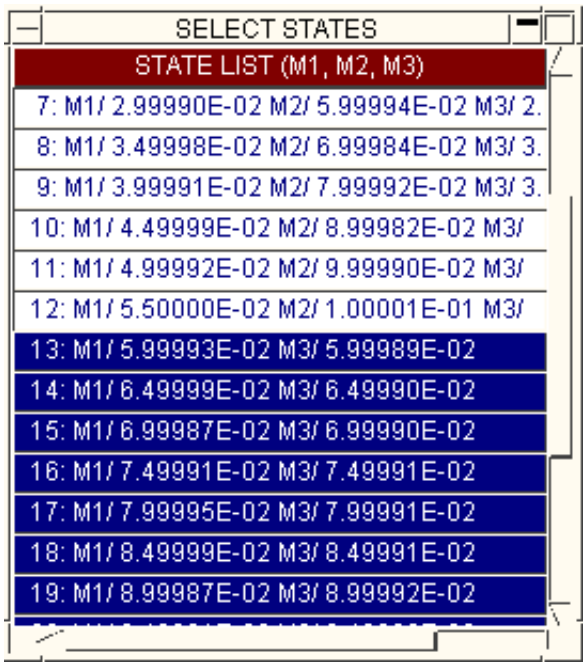
If they do not have the same number of states, or their states have different time intervals, animations across multiple windows may fall out of step in terms of "true" analysis time.

The remedy is to animate by explicit time interval, rather than by state number. This is done by interpolating "By Time" as follows:

Use **ANIM > SET_STATES > Custom...** to map the detailed state selection panel:



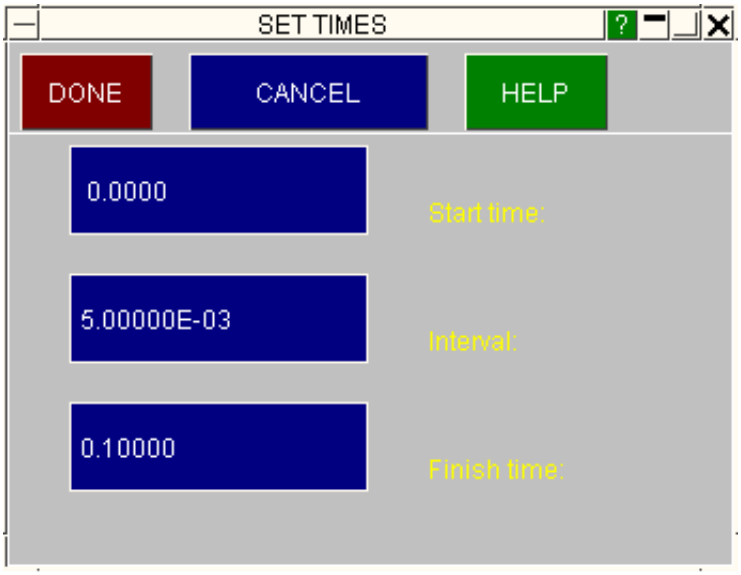
Then truncate the number of states in all windows so that they all have the same number of states. In this example it has been limited to 12states.



BY_TIME... Synchronising animation by time across dissimilar models.

The "by state" animation behaviour above is the default, chosen because it is simple and minimises memory usage, but it has disadvantages when models with different state time intervals are processed.

It is possible to interpolate between states and so to animate by user-defined time interval, which has the side-effect of synchonising models with dissimilar state times. This is done by using the **BY_TIME...** option. (Interpolation can be used on single models as well, usually to give a smoother animation, although this is less common.)



You define

```
<start time>
<time interval>
<end time>
```

And D3PLOT will interpolate states as required to achieve the specified intervals.

Interpolation is performed using a simple linear factor on the two "real" states that bound the required time.

You can control animation on a per frame basis using the slider and associated controls at the top of the window, exactly as described in section 4.4.1 above.

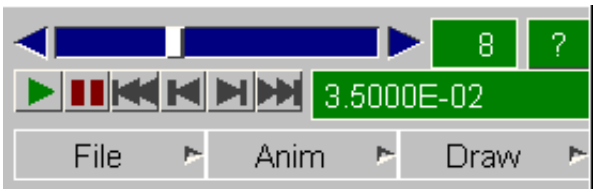
Controlling the display (statically) "by state".

To display the results (statically) at a "true" state while interpolating animations by time you need to revert to selecting the required state in the state control box.

This does not change animation back to "by state" mode, it simply reverts temporarily to showing the image at the time selected.

To revert to controlling animation by state, rather than by interpolated time, use any of the methods above to select states (eg **ANIM > SET_STATES**)

During interpolated animation by "time" the state box controls will always allow you to revert to showing a "true" state, not an interpolated one.



How interpolation by time affects output elsewhere in D3PLOT.

Interpolation by time affects the following other parts of the code:

- **WRITE** output will be given at the currently interpolated time.
- **DEFORM** options that use nodal coordinates (**SHIFT_DEFORMED, FIX_NODE, REFERENCE_NODE(s)**) will use the interpolated coordinates of the relevant nodes.
- **CUT_SECTIONs** that follow nodes will likewise use the interpolated coordinates.
- **UTILITIES, MEASURE** by node will report geometry from interpolated nodal coordinates.

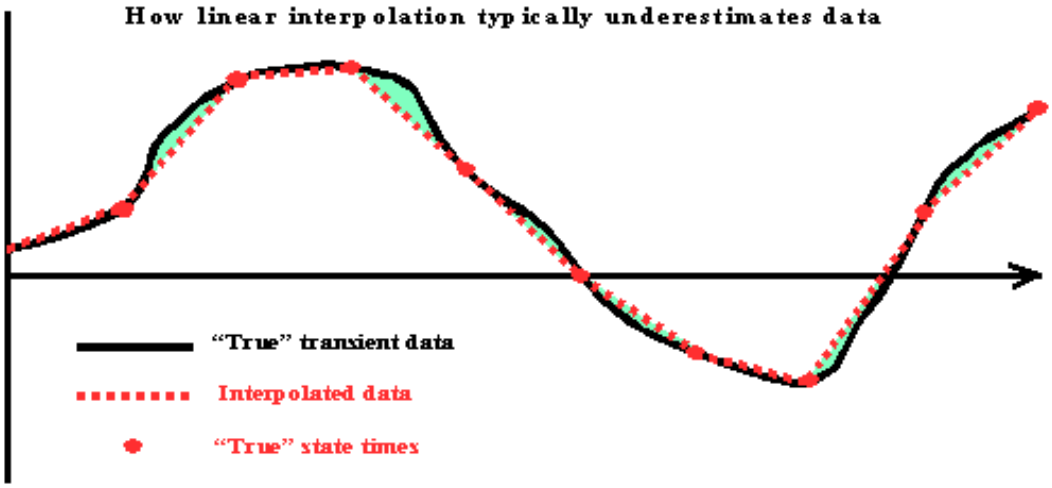
However note that interpolation by time does not affect the following:

- **XY_DATA** will still only report points at the time values of "true" states.

Warning: Interpolated values should be treated with care.

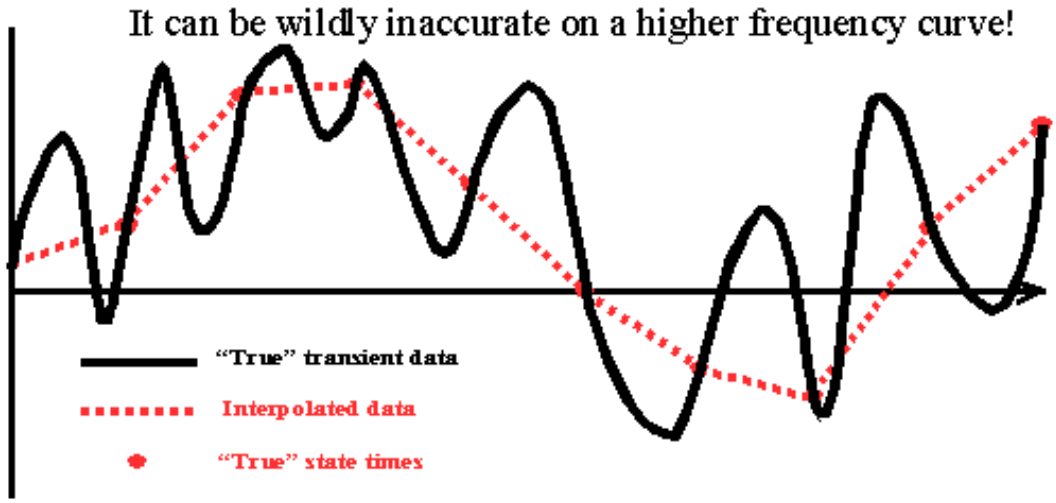
Linear interpolation tends to underestimate values between points on a smoothly varying, low frequency curve, and to give wildly inaccurate results on a curve which is high frequency relative to its sampling interval. This is acceptable for visual purposes during animation, but should be treated with caution if "written" values are to be extracted for subsequent use.

This is demonstrated by the following two plots:



Interpolation of a low frequency curve

In this case the green shaded areas show how the linearly interpolated results (red) underestimate the true values (black).



Interpolation of a high frequency curve

In this case, where the frequency of the curve is high relative to the sampling interval it is clear that linear interpolation (as well as the original points) will miss significant values.

Warning: Interpolation can be memory intensive.

Interpolation by time requires more memory (to hold the interpolated data for display) which, combined with the memory required to hold multiple models, can result in machine memory becoming exhausted. It is better to make sure that models being compared have near identical output intervals, as this greatly simplifies post-processing.

Animation "frames" (as distinct from states)

The situation can arise in which the images to be animated are not explicit "states" in the database. This happens when:

- Transient analyses are animated at interpolated times. (eg **BY_TIME...** as described above)
- A single loadcase of a static (ex-Nastran) analysis is animated in modeshape form.
- An eigenvalue (ex-Nike/Dyna or Nastran) modeshape is animated.

(For more on static and eigenvalue animation see Section 4.4.5)

To handle this D3PLOT has the concept of animation "frames":

- Each image in an animation is a frame, regardless of its origin.
- In the normal (not interpolated) transient case there is an exact equivalence between animated states and frames.
- In other (interpolated, modeshape) cases there will be at least two frames in an animation, but usually many more.

How does this affect you? Not very much, you only need to know the following:

- The slider and other positioning controls at the top of the graphics window operate on "frames" not explicit states, although in most cases these are the same.

  Therefore in cases where you are animating something other than **ALL_STATES** the effects of scrolling this slider (which navigates frames) and the state slider in the **State Display** box (which navigates states) will be different.

- You cannot stop an animation at a frame that is interpolated between explicit states.

  D3PLOT will not permit you to operate statically on interpolated frames from a transient analysis. This is because such results are potentially misleading: linear interpolation through non-linear data is inherently inaccurate. When you **STOP** an interpolated animation the current (static) state will be the one with the time closest to when you stopped the animation.

- Interpolated animations can be a bit slower than those at explicit states.

  Because you might choose a very large number of interpolated states D3PLOT does not store interpolated data for each frame, although interpolated coordinates are stored if space is available. Therefore there is a slight overhead as interpolated results are calculated "on the fly" during animation in these modes.

Click here for the next section

4.4.2.2 DISPLAY_MODE The display mode used for graphics

D3PLOT supports four possible display modes, which affect animation performance:

DIRECT Frames are calculated from scratch each time, no data being stored, so animation memory consumption is zero.

VECTOR The data to redisplay each frame is stored in "vector" form in the D3PLOT (client) process itself. This is the default.

PIXMAP **2D X-Windows only**. Each image is stored as a "pixmap" in the X11 server, no animation data being stored in the client process.

OBJECT **3D OpenGL only**. Each image is stored as graphical "objects" in the OpenGL server, no animation data being stored in the client process.

Why so many different modes?

The answer is the trade-off of replay speed vs. memory use, and the need to optimise this in some cases. For a small model this is not an issue, but as you start to approach the limits of your computer with larger models you may find that you need to alter the animation method, or even move to using two computers in client/server mode. The following table summarises this:

Display mode	2D X-windows	3D OpenGL	Comments
DIRECT	Speed: Slow Mem: Zero	Speed: Slow Mem: Zero	Use if memory is short and you can tolerate slow speed.
VECTOR	Speed: Fast Mem: Small	Speed: Medium Mem: Medium	Best all-rounder, set as the programme's default.
PIXMAP	Speed: Lightning Mem: Huge	*n/a*	50+ frames/sec until you run out of memory, then dire! Runs well on remote server.
OBJECT	*n/a*	Speed: Fast Mem: Huge	Again fast until you run out of memory, then dire. Runs well on remote server.

When do I need to change my display mode?

- If you get warnings about running out of memory during animation you may need to switch to **DIRECT** mode. Because of the way the operating systems on computers work it may be necessary to exit and restart D3PLOT to clear memory usage, then switch to this mode before rebuilding animations. Advice is given on-line about this if memory usage warnings are issued.
- If the default mode (**VECTOR**) is not fast enough, and enough memory is available, you can switch to **PIXMAP** (X-Windows) or **OBJECT** (OpenGL) mode.

What is meant by "runs well on a remote server"?

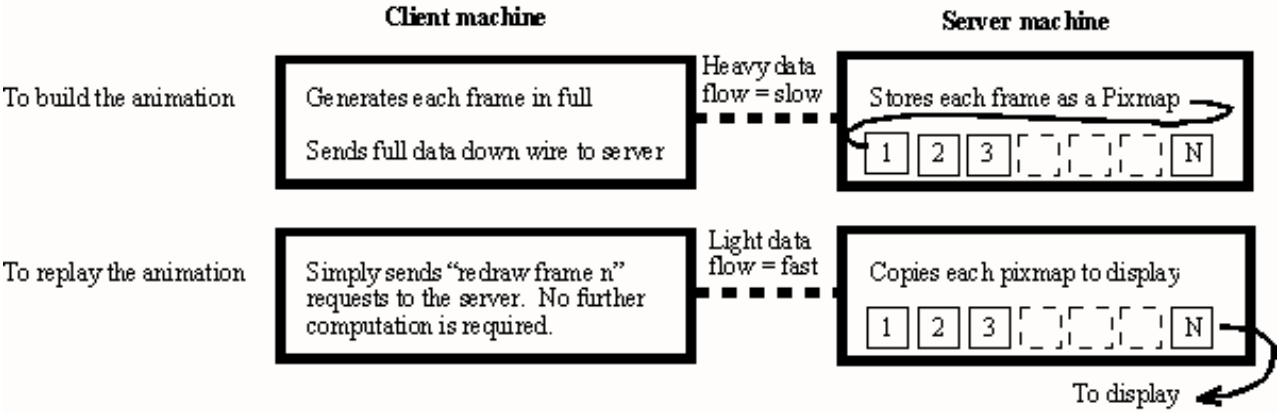
Under both X-Windows and OpenGL under X11 you are in fact using two processes:

- The "client" is the D3PLOT session itself, computing what is to be drawn. It passes these drawing requests to ...
- ... the "server", which is the process on the machine responsible for turning drawing requests into raster images on the display.

On a single machine the data transfer between the processes is fast - usually via shared memory - and OpenGL usage is generally direct from D3PLOT to the display card.

However there are some obvious advantages to separating the tasks between two machines: the client process can use all its host machine's resources to generate graphical requests, while the server can devote all its resources to displaying them. The disadvantage is that the data has to go down a piece of wire between the two machines, which may slow down data transfer. Therefore the best of all worlds is to store the graphics data in the server, whereupon the client only has to send a "draw this frame" request to render each frame of an animation.

Both **PIXMAP** and **OBJECT** display modes achieve this end in related, but different, ways.



PIXMAP mode (2D X-Windows only)

Is no slower to generate than **VECTOR** mode.

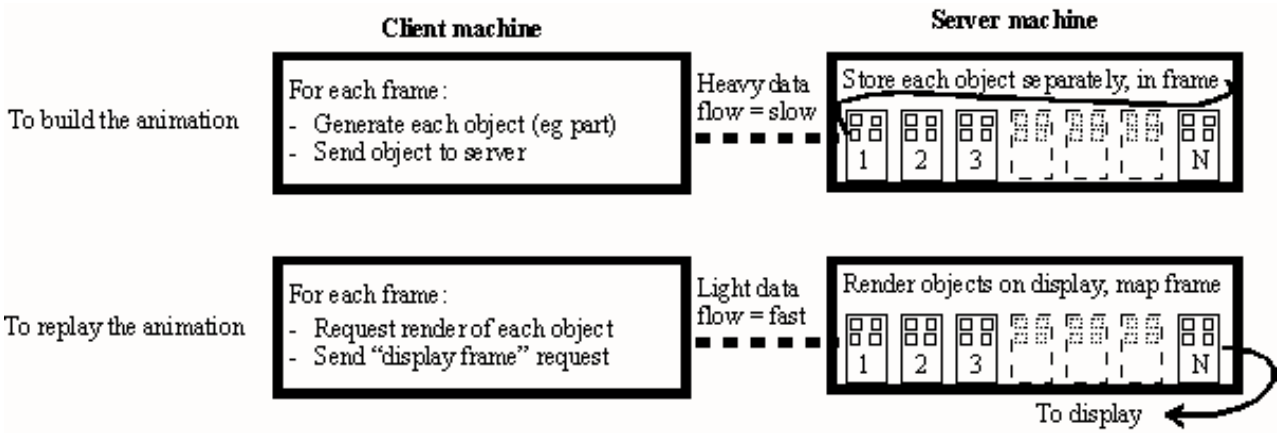
Its replay rate can be phenomenally fast, (> 50 frames/second is common), and is independent of image complexity once the frames have been assembled. Smaller windows give faster replay rates and also use less memory.

Memory requirements and replay rate depend on window size and visual depth. Memory required for each frame predictable: ([height x width x depth] bits / 8) bytes.

If you run the server out of memory it will slow down drastically, and ultimately lock up or even crash. D3PLOT sets an initial limit of 32MBytes for Pixmap memory usage, but you can alter this limit, or choose to resize the window to fit within it.

Being 2D any change to image content or viewing transform will require all frames to be rebuilt and re-transmitted to the server from scratch.

OBJECT mode (3D OpenGL only):



Is no slower to compute in the client than **VECTOR** mode, but can take longer to draw during the first pass because of the overhead of building "objects" in the server.

Is reasonably fast to redraw (usually about 1.5 - 2x **VECTOR** speed), but this is a function of the number and complexity of the objects that make up the image.

Its memory requirements in the server are unquantifiable but large - typically up to 10 times that required for **VECTOR** mode.

Since it cannot tell how much memory the server is using D3PLOT is unable to protect you from the ill-effects of excessive memory consumption there. Ultimately it may lock up or crash if driven completely out of memory.

In 3D mode a viewing transform change imposes no speed penalty. All that is required is to send a new transformation matrix to the server.

If the contents of an object change then only that object has to be re-computed and re-sent to the server. So the "cost" of changes is proportional to the extent of the changes made.

Usually the memory consumption of **OBJECT** mode, and its less than phenomenal speed advantage over **VECTOR** mode, make its use impractical if both client and server processes are on the same machine. However if you have two machines available, or you are working from a remote host and want image transforms to be quick on your local machine, it is worth trying.

Setting up a remote client/server connection: (This is only possible on machines running X11, ie Linux or Unix hosts)

A description of how to display on a remote server is given in Sections 2.1 to 2.4, but briefly:

Prior to running the **`oasys`** shell to invoke D3PLOT:

On the client machine:	Set the **`DISPLAY`** environment variable to point to **`<display>:0`** on the server. (Eg **`setenv DISPLAY server_name:0`**)
On the server:	Make sure that windows can be opened by remote clients. (Under Unix the command is **`xhost +`**)

4.4.2.3 **REPEAT >** The number of times an animation is cycled through.

By default an animation is repeated continuously until you stop it explicitly.

You can limit the number of passes through it to the options here, or to any number of your own choosing using the **Custom...** option.

If you limit the number of passes each **PLAY** request will result in that number of passes only. (The counter always starts afresh, it doesn't "remember" how many frames it displayed last time.)

4.4.2.4 **DELAY >** Delaying playback speed to achieve an explicit number of frames/second.

By default an animation is replayed at the fastest speed that the computer can manage.

Sometimes, especially in **PIXMAP** animation mode, this can be too fast and some frames get skipped. Alternatively if you are running multiple D3PLOT sessions, and you want animations to proceed simultaneously in several windows, you may find that you need to set an explicit display rate to stop one process "racing" ahead of the others.

Therefore it is possible to specify how many frames per second are displayed using the preset definitions here, or by using the **Custom...** option to select any frame rate.

Limitations of controlling playback speed.

On most computers it will be difficult to achieve ***controlled*** frame rates faster than about 60 frames per second since 60Hz tends to be the resolution of the average computer clock, and finer timing is not achievable.

In addition the refresh rate of your display is significant. Most liquid crystal displays (LCDs) run at 60 frames per second (60Hz), and typical cathode ray tube monitors at between 60Hz and 100Hz. Attempting to animate at rates faster than this not worthwhile, and can be counter-productive.

For small models it is possible that D3PLOT will deliver the frames at a rate faster than the display refresh speed, in which case one of two things will happen.

1. If the graphics adapter has "wait for vertical refresh before swapping buffers" set then the animation rate will peak at the display's refresh speed, giving good results. Signs of this are smooth animation and the cpu usage of the D3PLOT process dropping as it waits before swapping buffers.

   This is the best outcome you can achieve and you need not take any action.

2. If the graphics adapter does not have "wait for vertical refresh" set the results can be "tearing" between successive frames as an image is redrawn part way through a buffer swap and is thus made up of data from more than one frame.

   If this occurs then you should turn on "Wait for vertical refresh" on your graphics adapter.

   On Windows platforms this is usually achieved by ***<Right click on background>*, Settings, Advanced, *<graphics adapter name>*** and hunting through the options until you find the right setting.

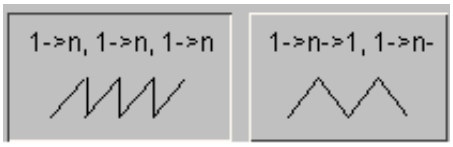
   On Unix / Linux platforms it is more difficult, and you may need to consult your hardware supplier for help.

4.4.2.5 **STYLE >** Setting playback to "sawtooth" or "modeshape" styles..

For transient analyses a "sawtooth" (1 => n, 1=>n) display generally looks more intuitive since it gives the impression of "start to finish".

However you can opt for "modeshape" (1 => n =>1 => n ...) mode if you wish (this is the default for static and eigenvalue analysis types).

Examples of the two different modes (from their **Custom...** panel) are shown here in diagrammatic form.



4.4.2.6 **DATA_SCAN >** How states are scanned to find max/min "automatic" contour levels during animation.

When you perform an animation of data with contours in "automatic" mode D3PLOT has to scan through all candidate states to find their max and min data values so that it can set the contour levels.

If you have a lot of states this can be slow, and in many cases you will know that your data values rise (or fall) monotonically, and that using values from the first and last states only will bound all possible values in between. Therefore you have a choice of:

EVERY STATE	This is the rigorous approach: every state is checked, and the true max and min values will be found. It can be slow if there are many states.
FIRST/LAST	Only the first and last states selected for animation are scanned for their max and min values. This is the default.

In order to protect you from missing peaks and troughs if, during the assembly of an animation, a data value outside the expected max/min values is found while in **FIRST/LAST** mode you will be warned and offered the chance to swap back to **EVERY STATE**. However if you do swap back it will be necessary to rebuild any frames that have already been computed in order to make them have contour bands representing the new max/min values.

4.4.2.7 **SAVE/LOAD** Saving animations to disk file. (X-windows only)

The use of this feature is deprecated: it only applies to 2D graphics using X11, and far better results will be achieved by saving AVI, GIF or MPEG animations as described in section 7.2

Click here for the next section

4.4.3 Improving animation performance.

For small models this will not be a problem, but as the model size and number of states grows so you will see that animation performance degrades. This section describes how to speed up animations by reducing the load on your machine.

The key to fast animation is to reduce picture complexity, (simpler images have fewer vectors and so draw faster), and to reduce memory consumption (forcing your machine to page-fault with virtual memory usage will cripple its performance).

Under X-Windows the use of 'pixmap' based animations will normally have a big effect on the animation performance since, once the pixmaps have been assembled, the replay rate is independent of image complexity. If 'pixmap' animations are being used then all of the following will have no effect except Section 4.4.3.5.

4.4.3.1 Choose an appropriate display mode.

Clearly the time taken to draw each frame will increase in direct proportion to the number and complexity of the screen vectors used. Therefore you should aim only to display the minimum quantity necessary. The following table gives an approximate "cost" on a scale of 1 to 10 for the various display modes on X-Windows and OpenGL devices:

Display Mode		Cost under X-Windows	Cost under OpenGL
LI	**(LINE)**	2	1
HI	**(HIDDEN-LINE)**	1	3
CT	**(CONTINUOUS_TONE)**	6	6
LC	**(LINE_CONTOURS)**	4	4
VEL/VEC	**(VECTOR/ARROW)**	5	5
SH	**(GREYSCALE SHADED)**	5	$2^{(1)}$ or $4^{(2)}$
SI	**(SHADED_IMAGE)**	10	$3^{(1)}$ or $5^{(2)}$ or $10^{(3)}$
ISO	**(ISO_SURFACE)**	4	6
CL	**(CLOUD)**	1	1

Notes: (1) No wireframe hidden-line overlay.

(2) With hidden-line overlay.

(3) With gouraud shading is turned off. (See Section 4.3.7)

The differences between "cost" on 2D and 3D devices in the table above are because of the fact that in 2D the effort is put in initially when the images are computed, and redrawing speed is simply a function of the number of vectors. Whereas in 3D mode effort has to be expended every time a frame is rendered, although some functions are much faster because of the hardware assistance.

For example **HI** hidden-line are the fastest display mode in 2D since they contain the fewest vectors, but relatively slower in 3D since each frame must be Z-buffered every time it is drawn. In contrast **SI** shaded image is slow in 2D since many contour bands (each costing a polygon) are used, but fast in 3D when hardware gouraud shading is used.

4.4.3.2 Minimise contouring effort.

Contouring can be especially graphics intensive and memory consuming:

- Turn the contour resolution to **Medium** or **Low**;
- Use the minimum number of contour bands
- Turn off labelling of line contours in **LI** plots;
- Turn gouraud shading on for **SI** plots under OpenGL.

4.4.3.3 Reduce extraneous screen vectors.

Turn off any extra information that is not definitely needed:
- Element and node labels;
- Node symbols;
- Element local triads.

Consider simplifying the display of entities. For example each 3D spring spiral contains about 70 vectors:
- Use "line" symbols for springs and seat-belt elements;
- Turn off contact segment hatching (broken lines are slow to render);
- Use free-edge or no overlay on data plots (especially worthwhile under OpenGL);
- Use flat shading for **SH** greyscale plots on 2D devices (no benefit under OpenGL).

If you are using cut-sections consider instead blanking the "unseen" parts of the model.

4.4.3.4 Consider using 2D mode for large animations under OpenGL.

Users with OpenGL will normally continue to use 3D mode for animation since it permits dynamic viewing and clipping. However the amount of information that has to be stored for animations in 3D mode is typically 4 times greater than for 2D mode. Consider switching to 2D mode (**`View Box, M2D`** command) if other measures fail.

4.4.3.5 Under X-Windows use a smaller window, and a visual with fewer bit-planes.

Animation under X-Windows is achieved by writing the image to a "Pixmap" in memory, and then mapping it to the screen. Clearly this involves moving a lot of data: the product of screen width, screen height and screen depth.

So reducing the window size will help, and indeed it is noticeable on most X displays how much animation speeds up as the window size is reduced.

The "depth" of a screen is the number of bit-planes in the visual. Most X displays use an 8 bit-plane Pseudo or TrueColor visual, and there is no benefit to be gained from reducing this. But some high-performance displays may be set to a 24 bit-plane visual, which gives three times the depth resolution required by D3PLOT. (You will get this if you used **X24** mode instead of **X8** when you started the code - see Section 2.2)

You can see what the depth of the screen is by looking at the text window from which D3PLOT was fired up, it will contain a message like:

```
Acquired display: width =   1280
                  height =  1024
        default #bit-planes =    8
```

(This example shows an 8 bit-plane visual)

If you are using a 24 bit-plane visual it may improve animation speed if you exit D3PLOT and restart it with an 8 bit-plane one. Use the **X8** device.

4.4.4 Laser plotting during animation.

It is possible to save animation sequences in laser files. Each frame becomes a separate image, so multiple plots on a page are recommended, and the current frame number and time replace the header on the plot.

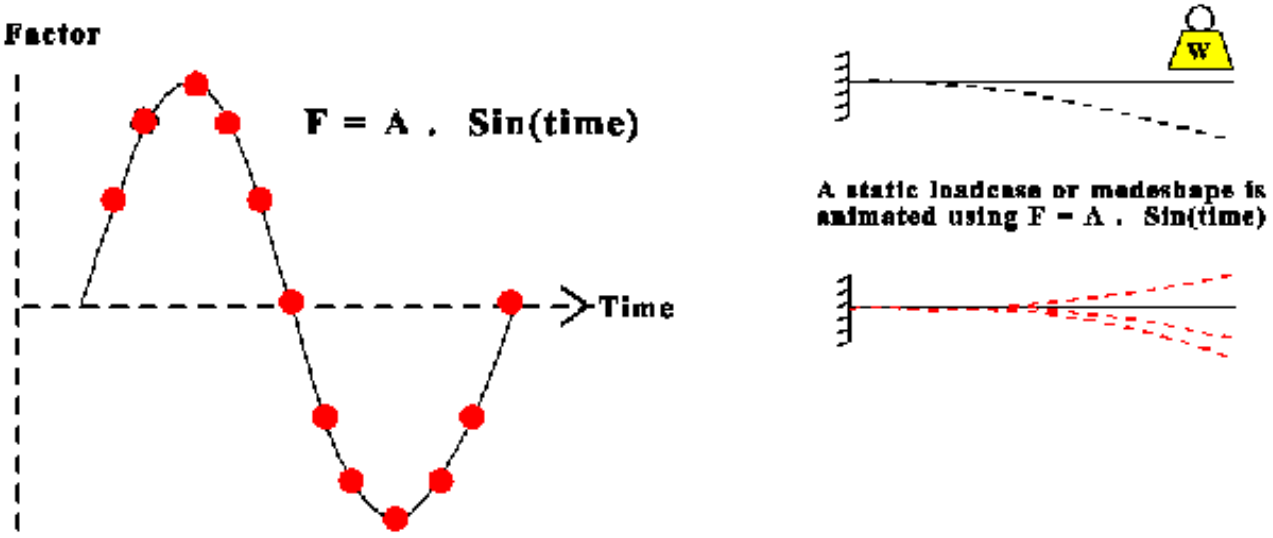
Laser plotting requires graphics to be in 2D mode on a 3D device. So if you are running under OpenGL in 3D mode you are warned that a switch to 2D mode is required, and you can choose to proceed or to abandon the laser plotting. (For laser plotting generally see Section 7).

4.4.5 Animating static and eigenvalue (modal) analyses.

Recent versions of LS-DYNA incorporate the implicit solver, and this means that they can generate eigenvalue results. In addition with suitable conversion software (contact Oasys for more information) it is possible to post-process static, eigenvalue and other solution sequences from Nastran analyses.

Analyses of these types differ from conventional transient analyses in that each "state" is assumed to be:
- Eigenvalue analysis: A given modeshape
- Static analysis:The result of a given static loadcase combination.

Therefore when such analyses are animated it does not make sense to animate over states, rather a given "state" is cycled through a sinewave function to produce a "modeshape" plot.

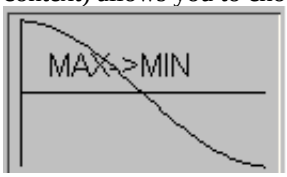


This has two implications for animation:

4.4.5.1 **ANIM >, SET_FRAMES** Setting the number of frames that are to occupy 360º of the sine wave.

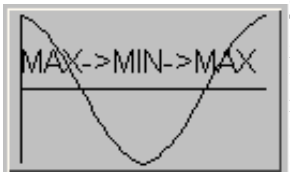
By default each 360 degree cycle of animation is split into 11 frames, which actually means 22 images, since the +ve and -ve cycles are symmetrical about their respective peak values.

The **SET_FRAMES** command in the **ANIM >** popup menu (which replaces the **SET_STATES** command in this context) allows you to choose a different number. More frames will give a smoother but slower animation.

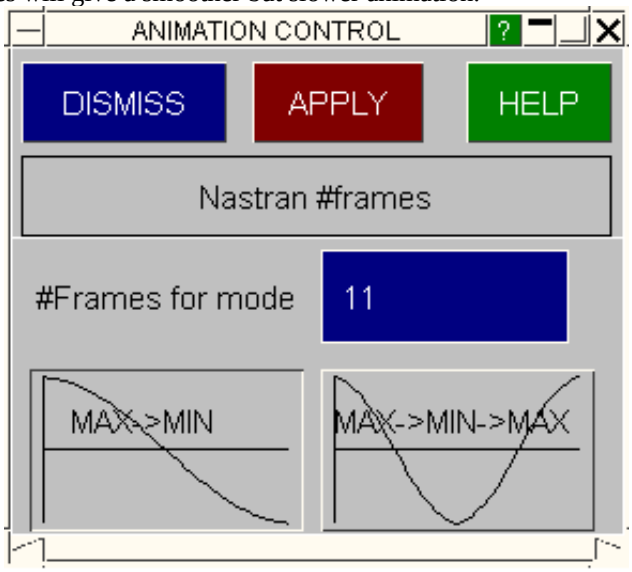


The **Custom...** option permits any number of frames to be defined, and also defines the period for the sine wave.

Normally the **MAX->MIN** option will be used, as this reflects the states internally to generate a 360^0 animation from 180^0 of frames.

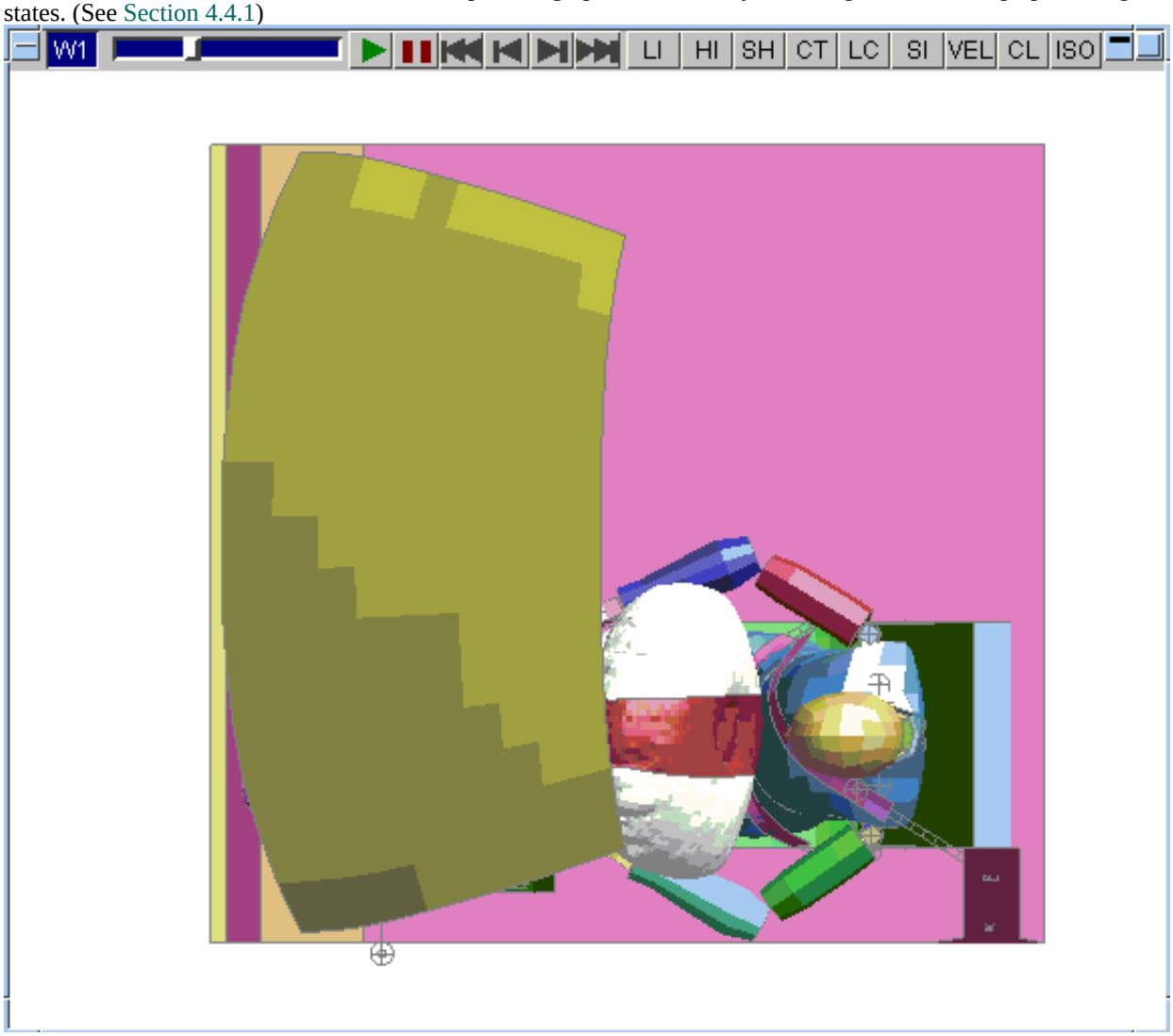


The **MAX->MIN->MAX** option is only required when generating files for an external viewer that is not capable of "reflecting" a 180^0 sequence into a 360^0 one. It looks stupid on the screen, but will duplicate the frames to produce a full 360^0 sequence in the file.



4.4.5.2 The frames slider cycles through the 0 - 360° cycle of frames, not through states.

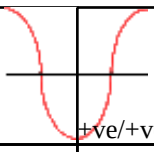
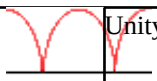
The frames slider, and other controls, at the top of the graphics window cycle through the modeshape phase angle, not states. (See Section 4.4.1)



4.4.5.3 Factors on results when animated by "modeshape"

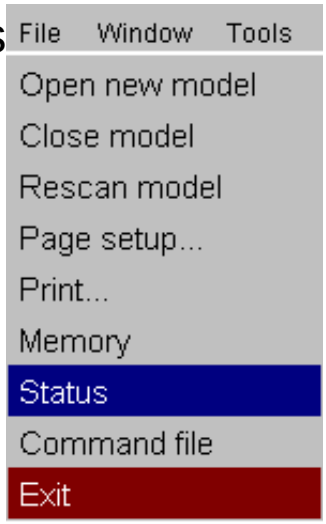
It is intuitively obvious that the factors on displacement to produce modeshapes need to be both +ve and -ve [factor = cos(time)]. It is less obvious what the factors on the corresponding results should be: magnitude values (such as von Mises stress) need a +ve/+ve, whereas direct stress tensor components (such a X direct stress) should be +ve/-ve, and components such as thickness should not vary at all.

Thus factors on data components through the 0 - 360deg vary as follows:

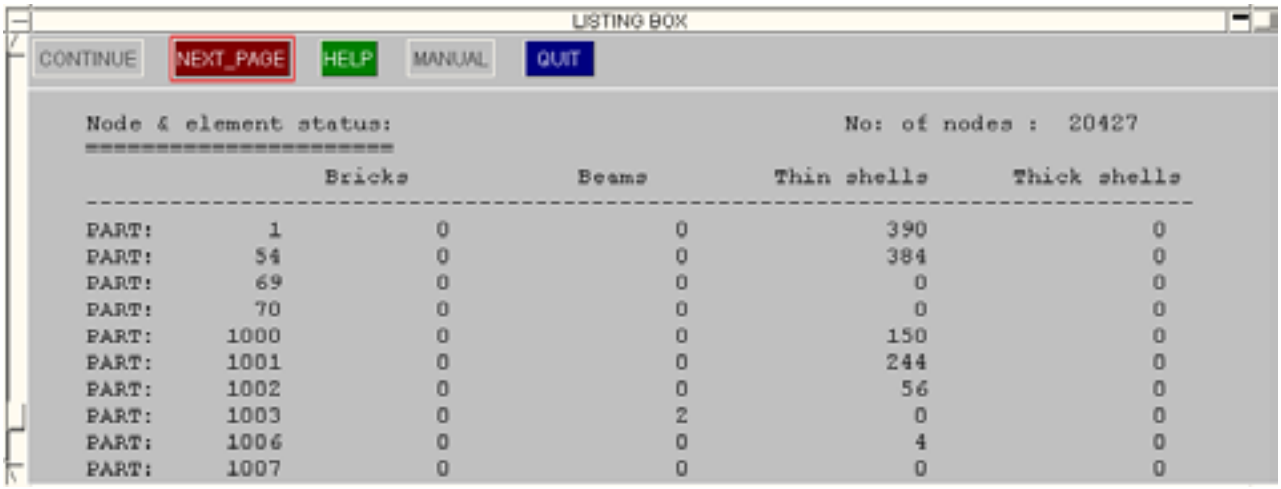
 +ve/-ve factors, f = cos(theta)	 +ve/+ve factors, f = |cos(theta)|	Unity factors, f = 1.0
[Sx,Sy,Sz,Txy,Tyz,Tzx] stress tensor [Ex,Ey,Ez,Exy,Eyz,Ezx] strain tensor Shell force & moment resultants [<outer fibre>] derived stresses [X,Y,Z] displacements	Everything not in the other two columns.	Thickness Shell Area Volume Outward normal Basic [X,Y,Z] coordinates Current [X,Y,Z] coordinates

Click here for the next section

4.5 **STATUS** Listing programme status



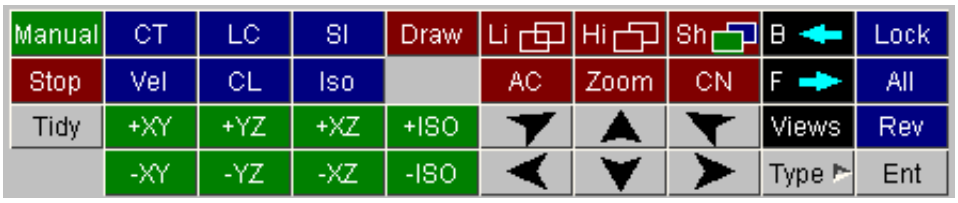
The **STATUS** command has no sub-menus or arguments, it simply lists the current programme status. The following figure shows a typical listing:


LISTING BOX
CONTINUE
NEXT_PAGE
HELP
MANUAL
QUIT
Node & element status:
No: of nodes : 20427
Bricks Beams Thin shells Thick shells
PART: 1 0 0 390 0
PART: 54 0 0 384 0
PART: 69 0 0 0 0
PART: 70 0 0 0 0
PART: 1000 0 0 150 0
PART: 1001 0 0 244 0
PART: 1002 0 0 56 0
PART: 1003 0 2 0 0
PART: 1006 0 0 4 0
PART: 1007 0 0 0 0

The listing gives a cross-reference of the number of solid, beam, shell and thick-shell elements in each material; a summary of other entity types; a listing of each contact-surface, and the overall model dimensions.

Lower down are also some key programme settings: blanking and volume-clipping switch status; the current in-core state number and time, etc. These are useful when D3PLOT is operating in command-line mode.

5 VIEWING CONTROL

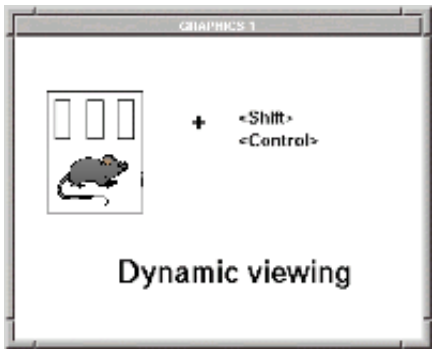


Controlling all aspects of viewing in the "Viewing Control" box

"Viewing" refers to the manipulation and presentation of images, rather than their actual generation. All viewing commands live in the "Viewing Control" box, located at the bottom right hand corner of the screen, and this section describes their use.

5.1 Dynamic Viewing (Using the mouse to change views).

"Dynamic" viewing is the name given to the process in which you perform viewing transformations by moving the mouse around the screen. This is the most useful way of controlling views.



5.1.0 Graphics modes during dynamic viewing

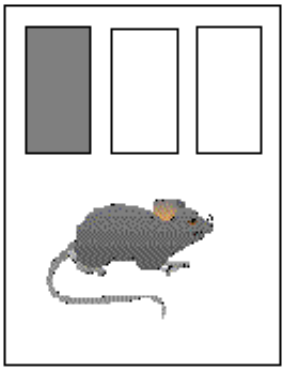
All dynamic viewing operations require a combination of two screen "meta" keys, (**<left control>** and **<left shift>**), and mouse buttons. The meta key(s) used dictates the graphics mode in which the image is transformed as follows:

<left shift>	+ **<mouse>**	Transforms the image in the current graphics mode. For example if it is a hidden-line plot, then dynamic viewing will take place in hidden-line mode.
<left control>	+ **<mouse>**	Transforms the image in "wire-frame" mode for the duration of the drawing operation. (ie no hidden-surface removal, or contours or lighting.)
<left shift> &} <left control>}	+ **<mouse>**	Transforms the image in pre-computed free-edge mode for the duration of the drawing operation. (ie wire-frame of free edges only, no hidden-surface removal, contouring or lighting.)

In the latter two cases the original drawing mode is always returned to at the end of the dynamic viewing operation. The wire-frame and free edge modes are provided to make transformations quicker for large models and/or slow computers: free edge is very fast.

For the last case, with **<left shift> & <left control>** held down together, the order of pressing and releasing the meta-keys matters: press **<left shift>** before **<left control>**, and release in the opposite order, otherwise you will (correctly) get the image redrawn in wire-frame mode as the **<left control>** key is pressed and released.

5.1.1 Dynamic Rotation.



Dynamic rotation uses **<left mouse> + <left shift> &/or <left control>**
(The distinction between the keyboard meta-keys is explained in Section 5.1.0 above.)

Rotation always take place in the **screen** coordinate system, and may be about the XY axes or Z: this depends upon the starting position of the mouse. This is shown in the next figure:

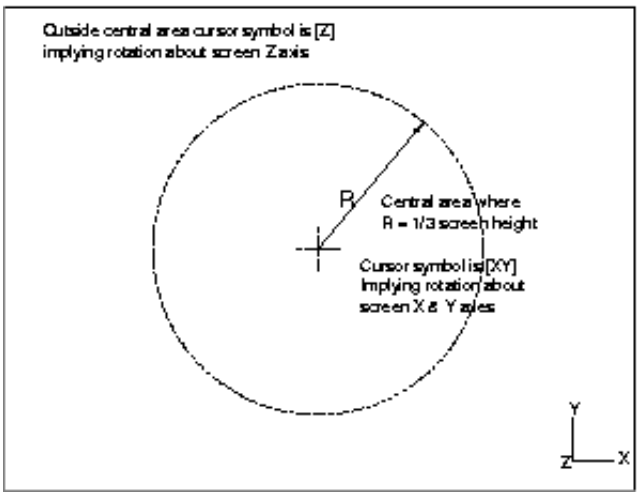
If the mouse initial position is **inside** the central circle (**radius (screen height/3)**) then rotation is about screen XY axes.

If the initial position is **outside** this circle then rotation will be about screen Z.

You can tell which mode you are in by the cursor symbol. This is **red**, and:

XY rotation uses **[XY]**

Z rotation uses: **[Z]**



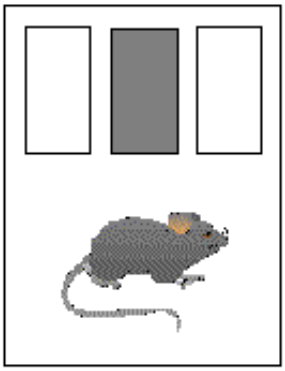
The relationship between mouse and image motion is intuitive in both modes. It is as if you had grabbed a point on the object near you, (this side of the object centre plane), and used this to move the image about its centre:

XY mode Moving the mouse left/right rotates about the screen Y axis;
Moving the mouse up/down rotates about the screen X axis.

Z mode Moving the mouse in a circular direction rotates about the screen Z axis.

Rotation remains locked in its initial XY or Z mode for the duration of a dynamic viewing operation, regardless of where you subsequently move the cursor to, until you release a mouse or keyboard button.

5.1.2 Dynamic Translation.



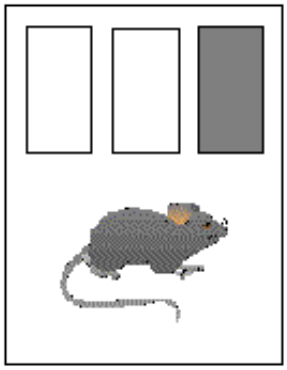
Dynamic translation uses **<mid mouse> + <left shift> &/or <left control>**
(The distinction between the keyboard meta-keys is explained in Section 5.1.0 above.)

The cursor symbol is **yellow**, and looks like:

Translation always take place in the **screen** coordinate system, in the X and Y directions.

The relationship between mouse and image motion is intuitive: the object tracks the mouse motion in the screen XY plane. The initial position of the mouse is irrelevant.

5.1.3 Dynamic Magnification (Scaling).



Dynamic scaling uses **<right mouse> + <left shift> &/or <left control>**

(The distinction between the keyboard meta-keys is explained in Section 5.1.0 above.)

The cursor symbol is **green**, and looks like:

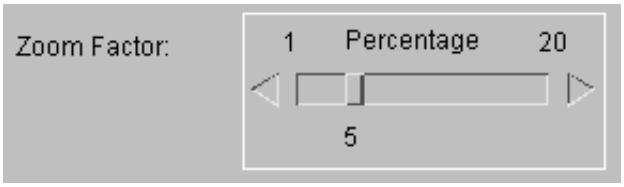
Mouse motion to the **right** and **up** makes the image larger, **left** and **down** smaller. The initial position of the mouse is irrelevant.

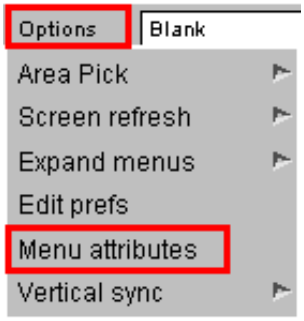
Dynamic magnification using the mouse scroll-wheel

If your mouse is equipped with a scroll-wheel then it will also perform dynamic magnification in the graphics window in which the cursor is present.

- Magnification is centred at the current cursor position unless "**CN** Centre node" has been used to lock centring on a node.
- Scrolling towards you magnifies the image scale.
- Scrolling away from you reduces the image scale.

By default each scroll wheel "click" will change the magnification factor by +/- 5%, but this can be changed using the **Options > Menu attributes** panel, and altering the **Zoom factor.**



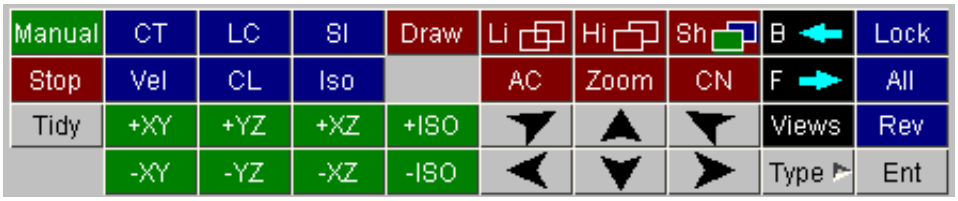


5.1.4 Dynamic viewing during animation.

On 3D devices operating in 3D mode you can carry out dynamic viewing during animation in exactly the same way as in static drawing. There should be no appreciable difference to the animation speed since all that is changed is the image transformation matrix.

On 2D X-Windows, (or 3D devices operating in 2D mode), you can also update the view during animation, but this will slow down animation dramatically as each frame will have to be recalculated at the new orientation.

5.2 Viewing Control Buttons

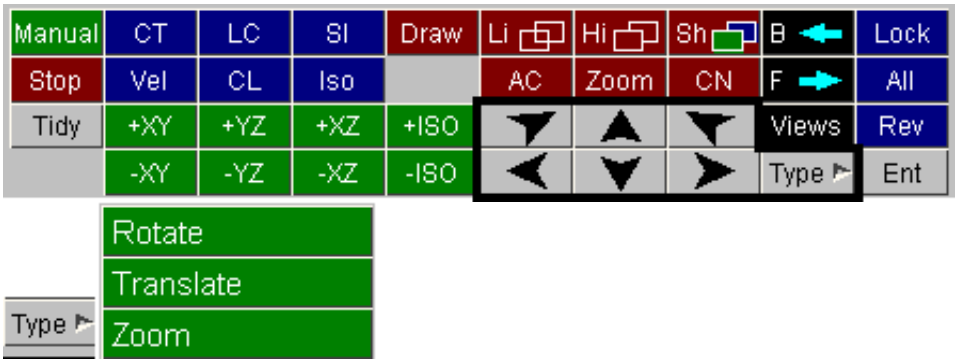


+XY, +XZ, +ISO etc	Pre-programmed view directions, also available from shortcut keys 1 through 8.
ZOOM	Zooms in by using the cursor to pick a rectangular screen area that is to be enlarged to fill the screen - also available from shortcut Z.
CN	Picks a node about which dynamic rotation occurs. This remains active (with the **CN** button lit) until disabled by pressing **CN** again.
AC	Calculates the correct scale and centre position required to make the current image fit neatly onto the screen. This takes account of blanking, clipping, deformations, etc. Also available from shortcut button A.
B <= and F =>	D3PLOT maintains a "history" of the last 100 views. The "**B <=**" button toggles bakwards through these and the "**F =>**" one forwards through them.
Views	Access to View manager (see section 5.3.1)
Lock	Prevent currently blanked entities being unblanked.
All	Unblank all entities not locked
Rev	Reverse blanking - also available from shortcut R
Ent	Access to the entity panel.(see section 6.5) - also available from shortcut E.

Command-line commands are also available (e.g.):

RM 30 0 0 - rotate (about model x,y,z axes) 30 degrees about the X-axis.
RS 30 0 0 - rotate (about screen x,y,z axes) 30 degrees about the X-axis.

5.2.1 Using the “Compass Rose”

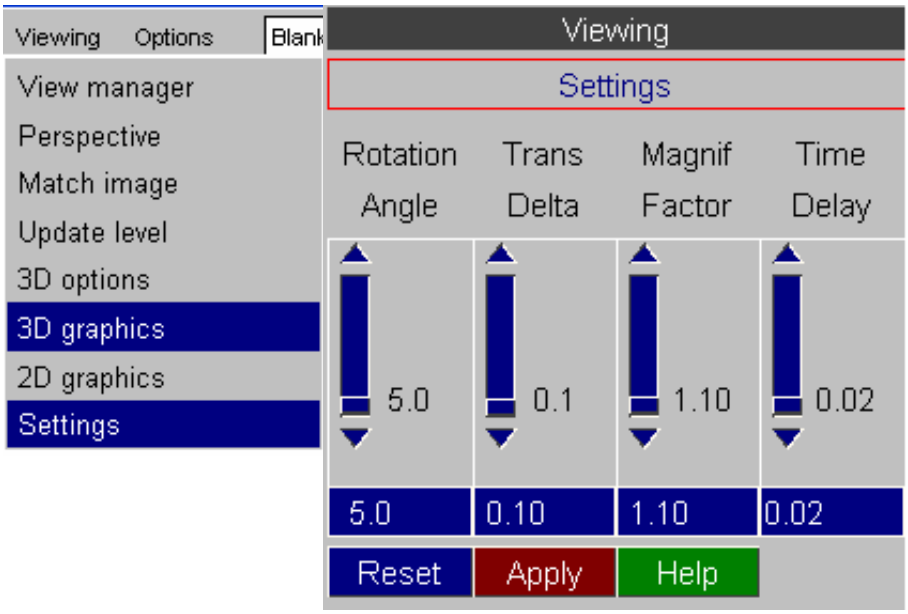


The "Compass Rose" provides three sets of buttons that allow the model to be rotated, translated and scaled with single mouse clicks.

Timed action of all of these is possible if buttons are held down, and the consequent repeated actions can be stored in command files making it possible to programme and record viewing sequences. Use the Type drop-down menu to switch between rotation, translation and magnification options.

5.2.1.1 **Setting compasss rose attributes**

This panel allows control of the magnitude of transformation per click, and the time delay bewtween frames when a button is held down.



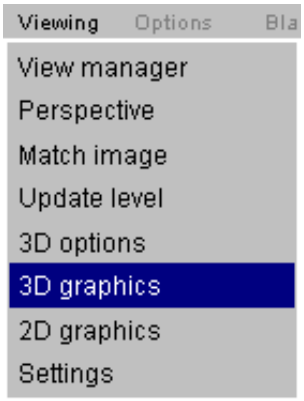
5.2.1.2 Programming transformations using the compass rose

The point has already been made that transformations using the compass rose are stored in command files just like any other command. It is worth repeating that "continuous" operations (those with a button repeating when held down) are also processed in this way: each repeat is stored like a separate "click" of the button. Thus a command file created in this way will have a sequence of many (possibly hundreds) of commands that are identical representing a series of repeated button presses.

You can use this to your advantage to make command files that rotate and/or translate the model automatically. For example you could pre-programme a "walk-through" of a structure in this way by saving the commands necessary to move your view point through and around and the model.

Click here for the next section

5.3 Options under Viewing menu



5.3.1 **VIEW MANAGER...** Storing and retrieving "view" information.

What is a view?

A "view" is all the information required to set up the current view of the object. In practice this means:

- The current rotation matrix (3 direction cosines).
- The current image centre location in space (x,y,z coordinate).
- The current magnification scale.
- The current perspective distance.

Up to 100 such views may be stored and retrieved at will from a file, and any number of such files may exist. The default file name is plot.view. A view is given a name and number when it is stored, and these are used when retrieving it. View files are binary compatible across platforms of the same word length (eg 32 bits), and are the same as those used by Oasys PRIMER.

Views are stored parametrically.

What this means is that views are not tied to a particular model, they will work for any model of similar dimensions. So if you are working on a set of variants of an analysis you can share the views on file between them: this is why they are stored in a separate, model-independent file. It is only when the shape and/or size of a model differs wildly from the original from which the view was created that this shareability fails.

Using views

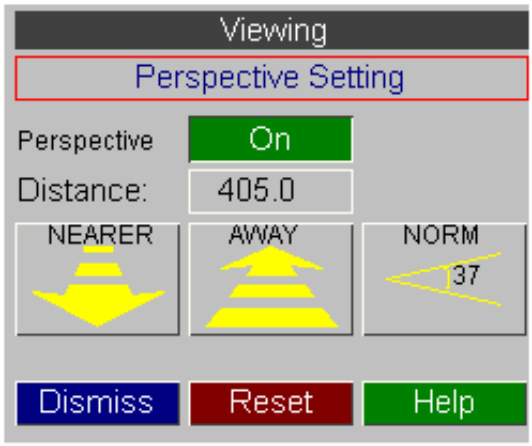
D3PLOT always has a current "view" definition. This dictates how the image will appear when a drawing command is issued. You can save the current view to file at any time. Likewise you can retrieve a stored view to replace the current one at any time.

The current view only exists in memory, and changing it has no influence on any views stored on file. (Indeed you don't need to have a stored view file: the default is none.)

Commands

STORE	Stores current view both in memory and in a view file. Click on a green (unused) view and type a name. Up to 100 views can be stored in a file, and views can be overwritten at will. If no explicit file has been opened the default file **`plot.view`** is opened automatically and used.
GET	Retrive from memory an existing view.
RENAME	Rename a stored view
DELETE	You can delete any existing vies
LIST	You can list information about stored views to screen
FILE	Define a file name in which views are to be stored

5.3.2 **PERSPECTIVE...** Setting Perspective Attributes.



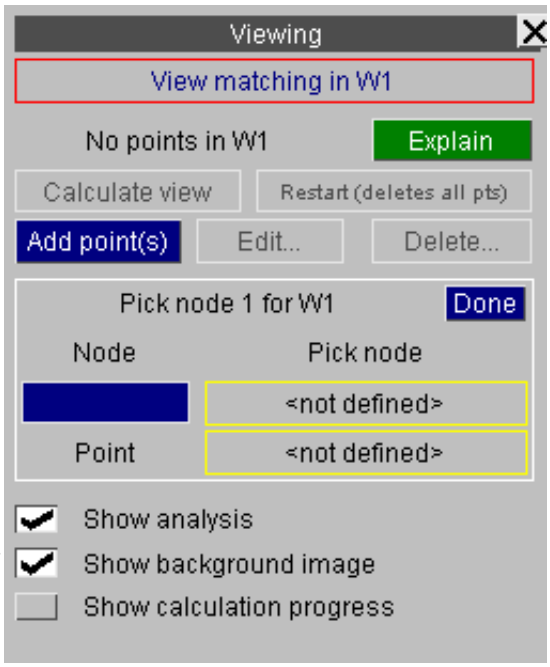
Use this option to switch on and adjust perspective settings.

D3PLOT will calculate the "bounding box" round your model and derive a default perspective distance of three times that value, which give a typical viewing angle of around 37 degrees.

Use **NEARER** and **AWAY** to adjust this, or type in a new **Distance** value.

If you get extremely close to the structure you may find that the overlay of hidden and shaded plots starts to come away from the underlying elements. This is a limitation of Z-buffered hidden-surface removal and a solution is given in section 4.2.2.2 under "controlling overlay quality in 3D mode".

5.3.3 **Match Image**



Automatically aligns the current analysis image with the background by calculating the transformation parameters required.

Lining up an image requires the calculation of 11 unknowns:

- The camera position (3 coordinates)
- The direction in which the camera is pointing (3 vector terms)
- The "Up" axis of the camera (3 vector terms)
- The distance of the object from the camera, ie perspective distance (1 term)
- The focal length of the camera lens, ie image scale (1 term)

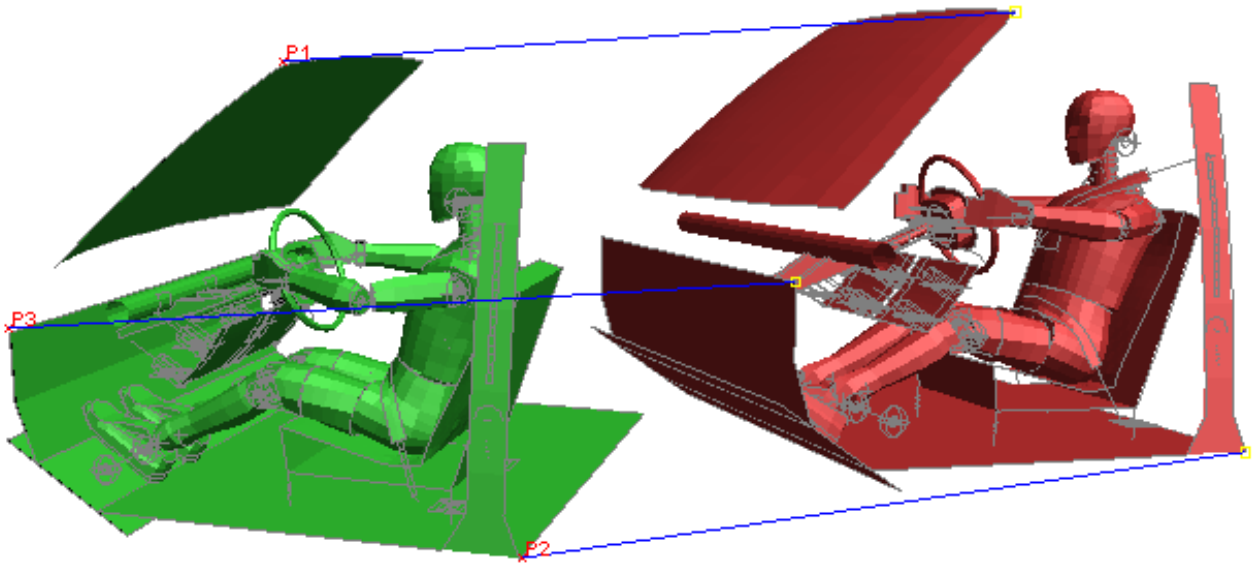
(In the orthographic case, where the object is viewed in a parallel sided frustrum, the perspective distance can be omitted leaving only 10 values to be computed.)

This calculation can be performed by D3PLOT if four or more nodes on the model are matched to their corresponding points on the image. Generally 5 or 6 points are required for a good match.

Add point(s) Defining <node : point> pairs for matching.

In the (artificial) example below the green image on the left has been read in as a background image, and the task is to get the red analysis image on the right to lie on top of it.

The user has defined 3 points so far: the nodes, identified by yellow pick symbols on the right, correspond to their matching points (red symbols and labels) on the left; the blue line shows which points and nodes are associated. These are screen-picked by selecting first the node, and then the corresponding point, and so on for the next pair.

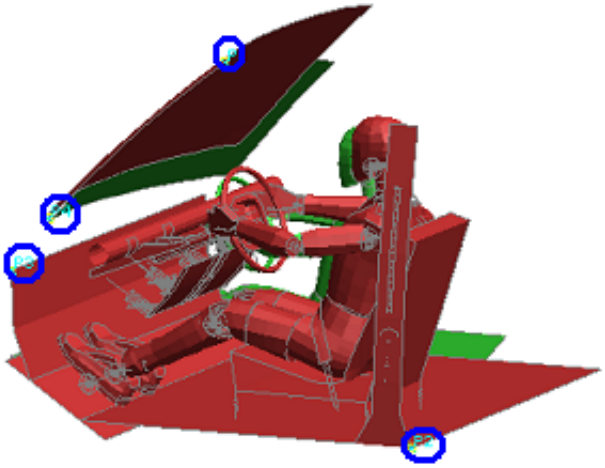


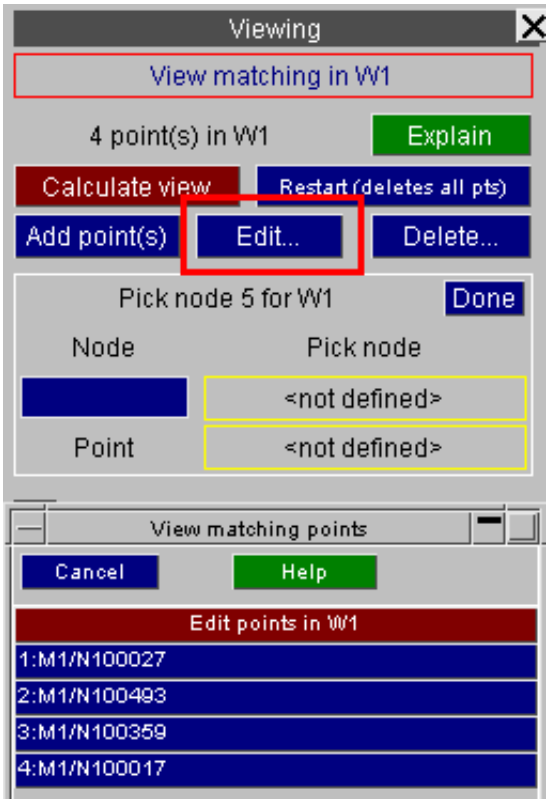
Calculate: aligning analysis with image.

Once four or more <node : point> pairs have been defined it is possible to calculate the revised view. This will calculate the revised viewing parameters and update the image immediately. If the images can be matched and the points have been well chosen then the analysis should lie exactly over the target image.

Edit...: correcting poorly chosen points.

In the example below points have deliberately been chosen badly to obtain a poor match. (The error here is choosing points, ringed in blue, that lie more or less in a plane, making it difficult to calculate perspective distance correctly. In addition choosing only four points is often inadequate, and more can be required for a good solution.)



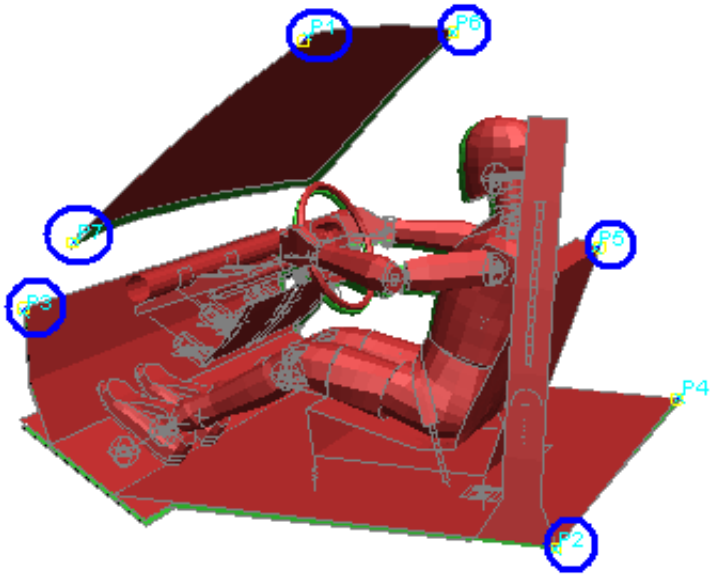


To edit a point screen-pick either its node or point (or select it from the menu), then repick its node or point.

Delete and **Restart**: Deleting points.

Delete allows you to delete individual points by selecting them as above. Each point is deleted immediately.
Restart deletes all points letting you make a fresh start.

You can **Add**, **Edit** and **Delete** points in any order. Here is the example above with 6 points (circled in blue) chosen rather more judiciously, and it can be seen that the correspondence is now very good.



What is stored for matching.

<Node : point> data is stored on a per-window basis, so it is not possible to apply matching data in Window #1 directly to windows #2, etc. However you can use the "Export view" function on the window's [--] options popup menu to export the current viewing parameters to all other active windows.

"Node" data is stored as a reference to a node in a model, and the current state's coordinate is used for matching purposes. Therefore if you need to match data during an animation you need to choose the state to be used for the matching process.

"Point" data is stored as a parametric (x,y) screen space coordinate, so points will remain valid so long as the aspect ratio of the window remains the same. However in most cases if a window is resized it is best to delete all the points and start again if further matching is required.

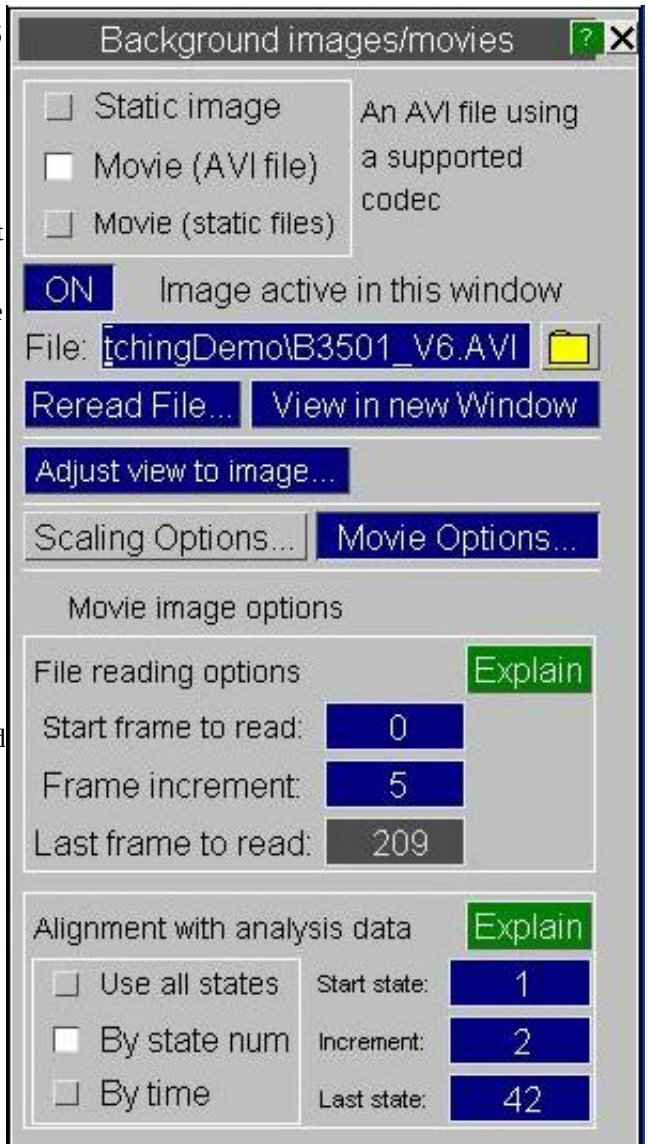
Matching a model to a series of frames of an animation.

At present image matching is "static". There is no provision for matching views separately to each frame of an animation. However, the model view can be matched to the first frame of the animation, which sets the model viewpoint at the real camera's position. Given that the camera's position is fixed relative to a known position throughout the animation, such as it moves with the model or is fixed to the ground, we can fix the model viewpoint, too in the same way.

In cases where the camera moves with the model you can use **Deform, Fixed Node** or **Shift Deformed** to track model movement.

In cases where the camera is fixed to the ground, you don't have to do anything because model viewpoint is fixed in the global coordinates at default.

After you have matched the model view to the first frame using the same technique as matching it to the background image, you need to match the timing, too. In this example we have a film with 0.002s per frame and a simulation analysis with 0.005s per state. To synchronize them we need every 5 frames of the film and every 2 states of the simulation analysis. You can set this at Movie Options.



5.3.4 **UPDATE Level...** Controlling the View updating frequency.

D3PLOT has an **UPDATE_LEVEL** setting which dictates how often the view is updated following commands that change it.

1:NONE	The plot is never updated automatically. Changes only become apparent when you issue an explicit drawing command, eg **DR**, **CT**, etc.
2:MEDIUM (default)	The plot is updated immediately when any view control command is given, or any quick-pick command. The current image is amended as necessary following blanking, clipping, etc if any viewing command, **including dynamic viewing**, is used. In other words a viewing change command is tantamount to an explicit redraw command in the current mode which would, of course, reflect any changes in the model geometry.
3:FREQUENT	The plot is updated immediately as at level 2 above, but also following any menu-driven blanking, clipping, etc, command that would change the image if explicitly redrawn. Therefore the effects of blanking, etc are seen immediately.

Note 1: Level 3 is only recommended if you have a very fast display and/or a small model since it requires frequent redraws.

Note 2: Users with slow devices and/or with large models may find that level 1 is preferable to decrease redrawing effort.

Click here for the next section.

5.4 Special 3D graphics driver options (OpenGL).

Most of the information in this section is included for backwards compatibility. 2D graphics are only rarely used nowadays, and the capabilities of the 2D graphics modes in D3PLOT are no longer being developed.

5.4.0 Brief description of 3D vs. 2D graphics.

In 2D mode D3PLOT treats the display device as a dumb 2D device on which lines, polygons and text can be drawn. All coordinates are expressed in 2D integer space, ie [x,y] only, and all calculation of hidden-surface removal, lighting, contouring, etc must be done in software. Dynamic viewing relies on the sofware recalculating and redisplaying images quickly.

In 3D mode much more intelligence is available in the graphics driver, and much of the effort of computing images can be shifted from the software to the hardware. In particular:

- Graphics coordinates exist in 3D [x,y,z] space, and the hardware does the transformation and projection onto the 2D screen. The software only has to provide the raw coordinates for an image once, and thereafter to change the view only a new scale, centre and rotation matrix.
- The hardware can compute shading, lighting and hidden-surface removal. So, again, the software only needs to provide raw coordinates, topology, light source data, etc, and then just ask the hardware to render it.
- The hardware can provide functions, such as Z-clipping, that are not available in softwar

So 3D devices, especially those with hardware acceleration, give much faster graphics.

However there are also drawbacks to using 3D graphics: more memory is required since the full scene has to be sent to the driver using [x,y,z] floating-point coordinates, and this can cause problems during animation when many frames have to be stored. In addition laser plots cannot be generated by the 3D driver, so the capability to switch temporarily back to 2D mode has to be preserved.

Therefore there are options to control aspects of 3D graphics, and also the ability to switch back and forth between 3D and 2D modes.

5.4.1 **3D Graphics, 2D Graphics-** Switching between 3D and 2D modes.

You can switch explicitly between 2D and 3D modes using the **M2D** and **M3D** buttons.

Some other graphics options also cause a switch.

On a 3D graphics driver the default mode is 3D, ie **M3D**, but certain graphics operations will switch the mode back to 2D. These are:

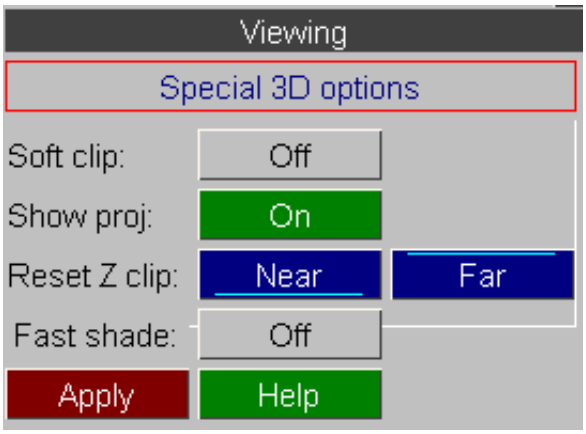
- Switching on dithering, either of contours or shading. See Sections 4.3.6 and 4.2.3. 2D mode continues until you switch it back explicitly.
- Plotting with laser output turned on. 2D mode is only transient during the course of the plotting operation, it

There are other circumstances when you might also want to switch explicitly to 2D mode:

- When producing **LC** line-contour plots the result in 3D mode can be a bit patchy, with contour lines dropping in and out of view. This is a function of the Z buffering in hardware, and software (2D) images look much better.
- When using the **OPACITY** switch for contact surface and beam plotting. This works after a fashion when in 3D mode, but the transparent structure overlay does not use proper hidden-surface removal. The results are better in 2D mode where more control is available in the software.
- When animating large models. The amount of data stored for a 2D animation can be far less than for 3D, and can get round memory shortage problems. (However you would do better to use the X-Windows driver in this situation: see Section 4.4.5.4.)

5.4.2 **3D_OPTIONS...** Further 3D options

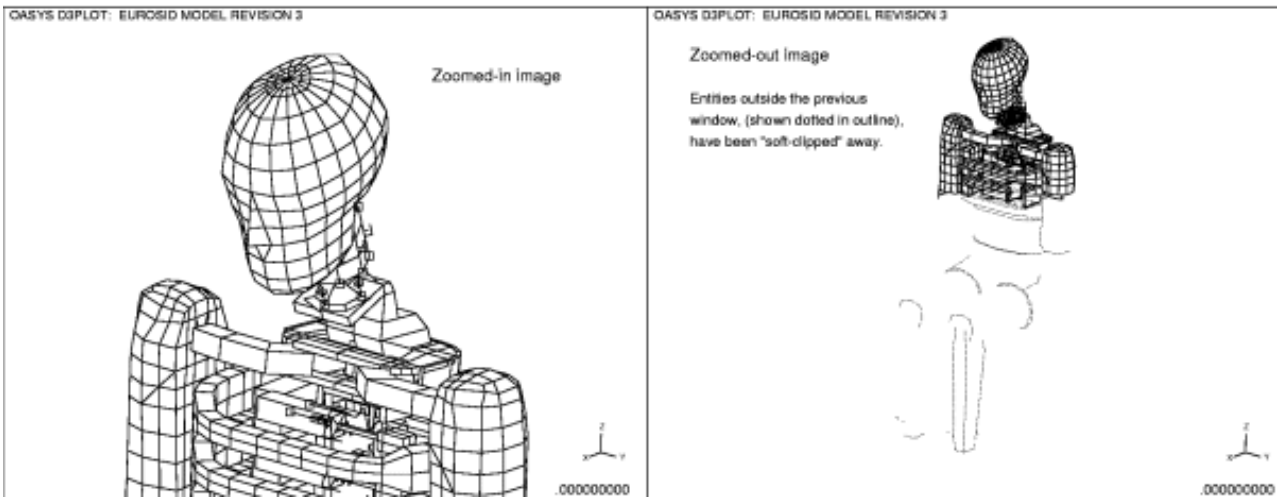
The **3D_OPTS...** button gives a control panel for further 3D options. These options are described below.



5.4.2.1 **Soft clip** Clipping graphics outside the current screen window.

If you are dealing with a very large model, but are only looking at a small part of it, the 3D graphics driver can work unnecessarily slowly in its default mode of operation. This is because the whole model is sent to and manipulated by the graphics driver, despite the fact that you are only looking at a small part of it, in anticipation of your wanting to zoom out to see the whole of it.

If you turn **Soft Clip** **on**, and redraw the image, the graphics will run faster. This is because the software has "clipped" (ie removed) those parts of the image not visible in the current window before sending it to the 3D graphics driver, so the 3D driver has to process fewer graphics entities. However this also means that if you zoom out those parts of the image outside the previous window will not be there. This is illustrated in the figures below.

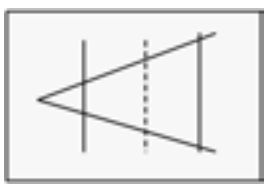

OASYS D3PLOT: EUROSID MODEL REVISION 3
Zoomed-in Image
.000000000
OASYS D3PLOT: EUROSID MODEL REVISION 3
Zoomed-out Image
Entities outside the previous window, (shown dotted in outline), have been "soft-clipped" away.
.000000000

In this example the user has zoomed in on the neck and upper chest region of a side-impact dummy (left hand image), and then zoomed out to what should show the full dummy. This exposes the jagged edges left by the 3D clipping algorithm.

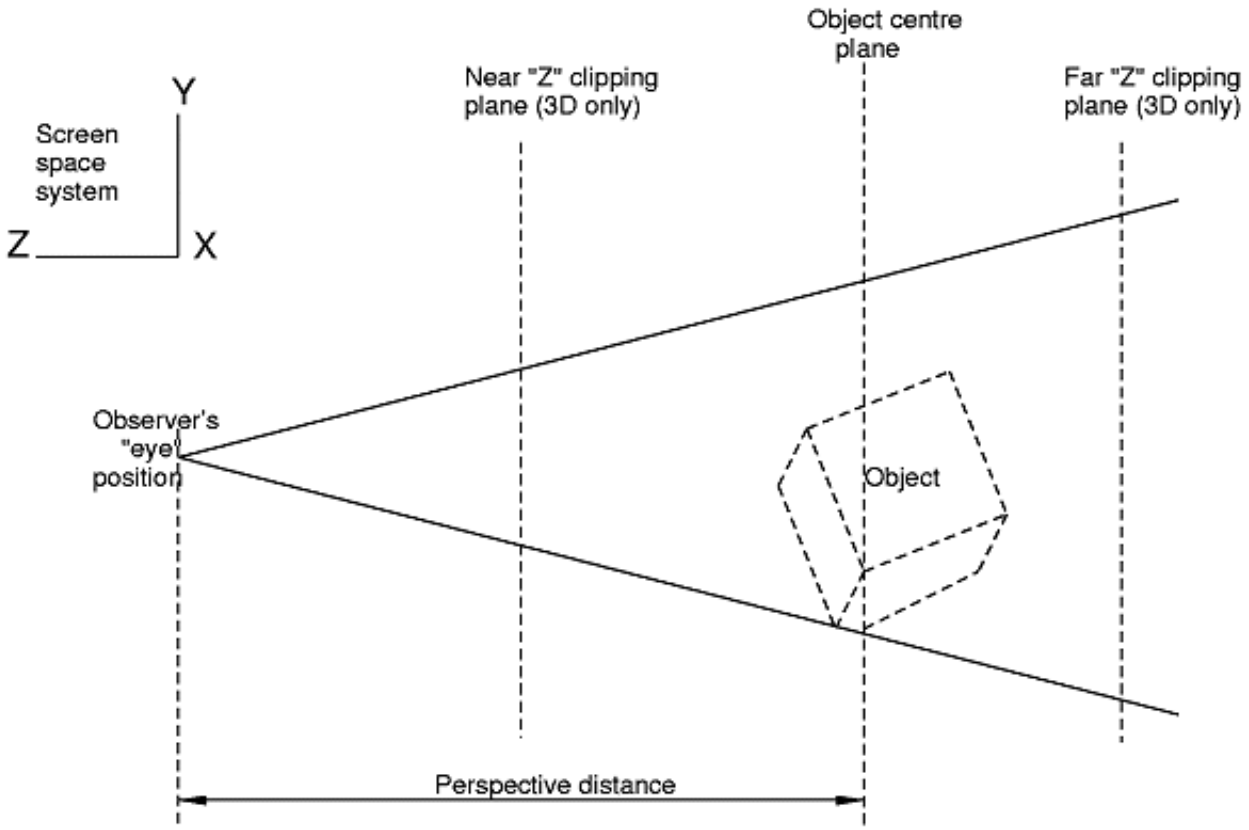
To see the missing elements you need to issue an explicit drawing command at the new scale to recalculate the clipping and send more elements to the 3D graphics driver.

5.4.2.2 **SHOW_PROJ** Showing the viewing frustrum

On 3D devices it is possible to show the current viewing "frustrum" at the bottom left corner of the plot by turning **SHOW_PROJ** on.



This shows the information in the figure below (a copy of The D3PLOT viewing frustrum figure in Section 5.1).



The frustrum shown here assumes perspective projection.

The Z clipping plane locations are shown when **SHOW_PROJ** is on, and this can be very helpful when using Z clipping, as otherwise it is easy to "lose" the clipping planes.

The default near and far plane positions are drawn in green, and the plane locations in blue. So you can visualise movement relative to initial locations.

5.4.2.3 Using the Z clipping planes

The Z clipping planes are shown in the figure above. There are two planes: a "near" and a "far" one, which the hardware uses to clip the image in the +/- screen Z axis.

By default they are set just outside the +/-Z limits of the structure (shown as green lines in the projection box), so that no clipping takes place, but when the 3D options box is mapped you can move them (shown as blue lines in the box) using the following mouse and keyboard meta-key combination:

<right shift> + <left mouse>	Moves the **near** clipping plane.	Cursor symbol is
<right shift> + <right mouse>	Moves the **far** clipping plane.	Cursor symbol is
<right shift> + <mid mouse>	Moves the **both** clipping planes.	Cursor symbol is

(Note that when the 3D options box is ***not*** mapped then the <right shift> and <right control> keys act exactly like their <left> equivalents, meaning that either side of the keyboard can be used for normal dynamic viewing.)

In all cases moving the mouse **up** moves the plane(s) **away** from you, and **down** moves **towards** you. This is a form of dynamic viewing: the planes move and the image gets updated as the cursor moves. It is recommended that you turn the **SHOW_PROJ** switch, described above, on as this will enable you to see the planes moving in the projection box.

To reset the planes to their default positions use the **Reset Z clip NEAR** and **FAR** buttons. This will reset them to their initial positions (shown by the blue lines in the projection box).

6 USING "TOOLS" OPTIONS

This section acts as a brief introduction to the commands in the top (**Main Menu**) box.

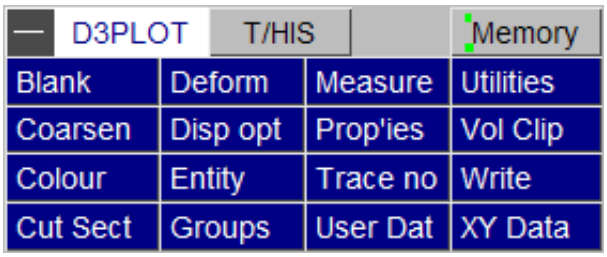
6.0 Introduction to main menu commands

6.0.1 Commands invoked from here are mutually exclusive

The commands in this box are mutually exclusive.

Panels for these commands are mapped in the fixed area below the main menu, stacked in order of invocation, and the button of the current command is highlighted.

Click on a command below to jump to its detailed description:

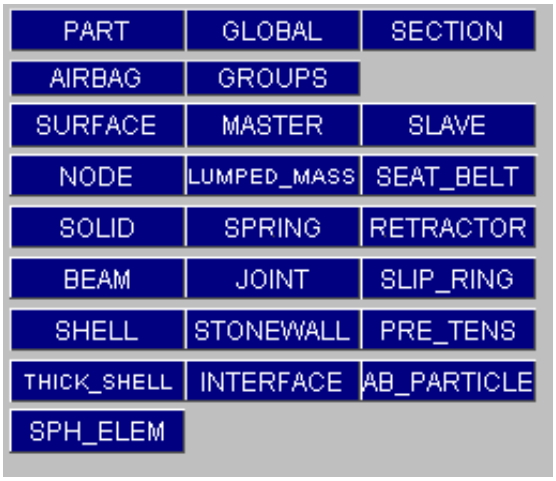


Blank	Deform	Measure	Utilities	D3PLOT	T/HIS	Memory
Coarsen	Display Options	Properties	Volume Clipping			
Colour	Entity	Trace nodes	Write			
Cut Sections	Groups	User Data	XY Data			

6.0.2 Using standard display list selections

Several of the functions in the main menu require you to input <lists> of entities to be processed. The standard procedure for this is:

(a) Select an entity type
This figure shows the standard entity type entry panel. All possible entity types are always mapped, with those that are unavailable greyed out.



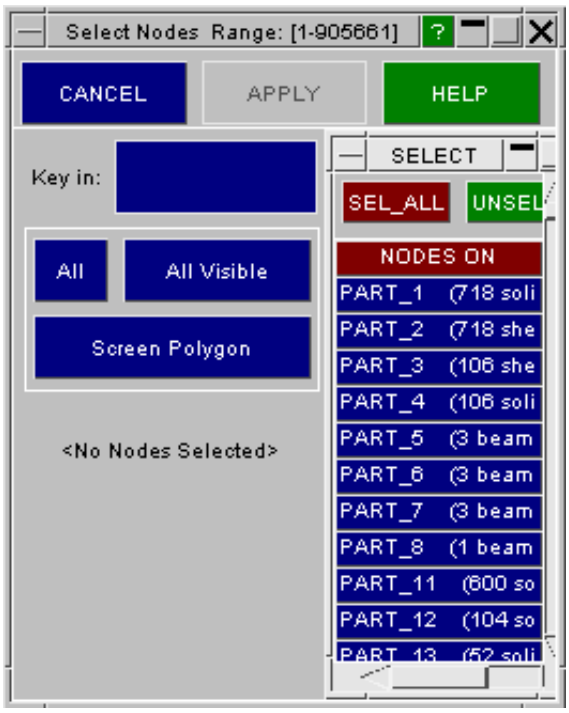
(This example is from a **[WRITE] ENTITY** command.)

In some contexts some or all of the **MATERIAL**, **GLOBAL**, **SURFACE**, **MASTER & SLAVE** options shown here may not be present since they are not appropriate.

(b) Define a <list> of these entities

After you have chosen an entity type you must define a <list> of entities to be processed

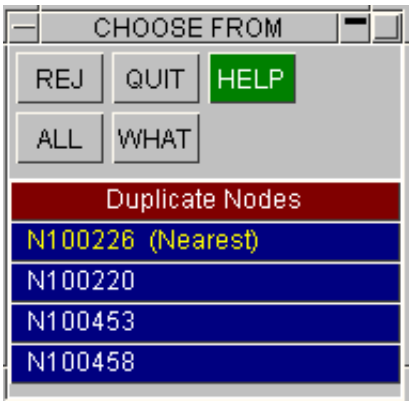
This figure shows a typical panel displayed after a command. You can select a immediately by clicking on or dragging across visible entities. Other options are:



Key in	To type in a range;
ALL	To select all entities;
ALL_VISIBLE	All currently visible entities;
SCREEN POLYGON	Pick points defining a polygon within which entities will be defined

Or select entities on material(s) using the menu (here headed **NODES ON**).

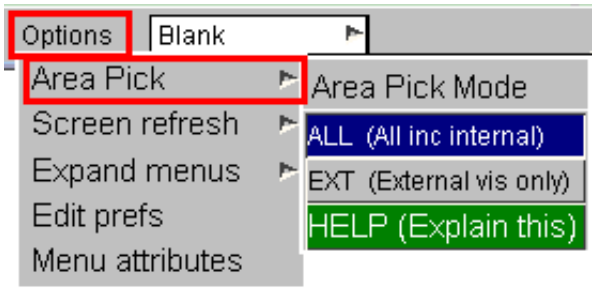
Resolving ambiguous screen picks.



When screen-picking you may occasionally pick a point that does not lie unambiguously on an entity. In this case the "ambiguous pick" menu, see the figure (right), will be mapped, and you will be forced to choose an item. You can:

- Choose a menu item: the top one is always nearest to the point you picked, or:
- **REJ**ect this pick: the pick is ignored and you get another chance.
- Take **ALL** items from the menu list.
- Show **WHAT** these items are by labelling them.
- Abort the whole picking operation using **QUIT**.

Treatment of 3D elements during Area or Polygon picking

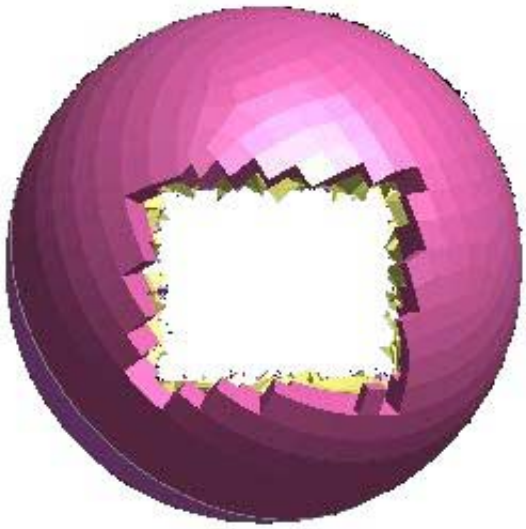


Options, Area Pick controls how 3D elements are treated during screen area type picking. (This does not affect single picks, which will always take the nearest element only.)

ALL All eligible elements in area

By default area picking of a mesh that contains solids or thick shells will include elements that are eligible for display, but which have not actually been drawn because they are interior to the mesh.

You can think of this as all elements in the "tunnel" behind the screen area: blanking in this way will punch a clear hole right through the mesh as shown in this example.

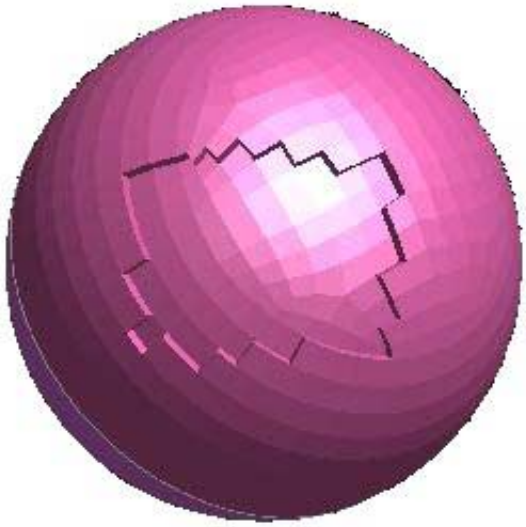


EXT Only external elements

Alternatively only those elements which have actually been drawn, for a 3D mesh the **EXT**ernal surface, will be selected: elements culled from the display because they are internal are not selected.

The effect of a pick in this mode is like peeling an onion: only the outer layer is removed in each selection pass.

Contrast this example with the image above: only the outer layer of solids has been removed.



6.0.3 Using command-line syntax for <lists> of entities

In some circumstances it can be much quicker to use command-line input (in the dialogue box) to define <lists> of entities. A typical example might be when you want to process an explicit list of known elements, or a well defined range.

Typical command-line syntax to perform an operation is:

```
Command (command) (command) <list of entities>
```

for example:

```
/WRITE NODES 21 to 100
```

The valid syntax for a <list> of integers defining a range of entities is:

Single values `12 20 200 -1`

a TO b (STEP c) `1 TO 100,  -20 TO 40 STEP 4,  1000 TO 100 STEP -2`

All in context ALL or *

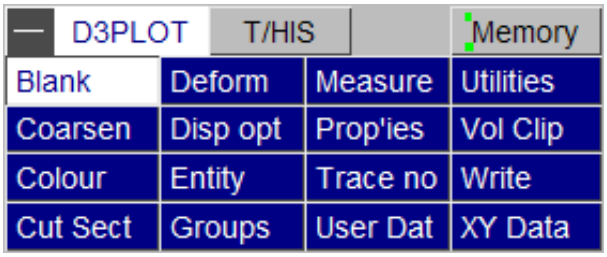
Range limits `FIRST` and `LAST`

All of these input types above may be mixed at will on a single line. Continuation lines, using ****, may be used in the same way as for command words.

You can mix screen-menu and command-line input at will.

Next section.

6.1 **BLANK** "Blanking" controls the visibility of nodes and elements.



You can cut down what is displayed by "blanking" nodes and elements. (Unlike **ENTITY** display control blanking is selective: you can blank and unblank individual elements.)

Each node and element in your model has an internal blanking flag, which is initially set to **off** (ie the entity is visible). You can turn this flag **on** by blanking that entity, and in subsequent plots it will not be drawn until unblanked again.

The BLANK panel operates on one model at a time.

If you have more than one model then you will be forced to choose which one the blanking panel operates on. You can subsequently change this by using the **`M1 .. Mn`** tabs.

Blanking is "per model", not "per window".

The blanking status of items is stored at the model level, not at the window level. Therefore if something is blanked in one window it will also disappear from any other windows in which it occurs once they are redrawn.

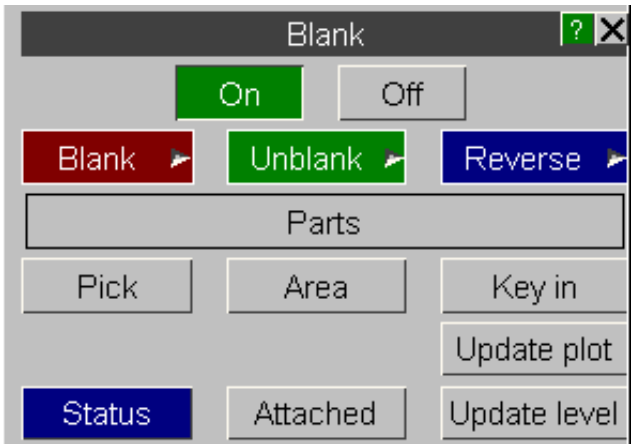
This is done for simplicity and to save memory. If you need to have two concurrent images of a model with different blanking attributes then you will need to read that model in twice, treating it as two separate, albeit identical, models.

DYNAMIC ("Quick Pick") BLANKING

The **BLANK** panel is used when you want to exercise detailed control over what is blanked. The alternative method of blanking, referred to as "quick pick" mode, which is more suitable for simple blanking is described in section 3.5.

The **BLANK** menu is split into 2 sections.

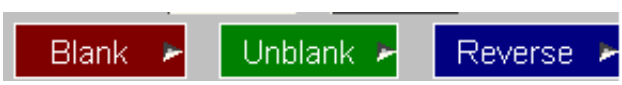
6.1.1 Selecting Items



The top half of the menu provides ways to select the items to be blanked and unblanked while the bottom half is used to choose the type of item that is going to be blanked / unblanked.

ON Turns blanking **ON** so that blanked items are not drawn.

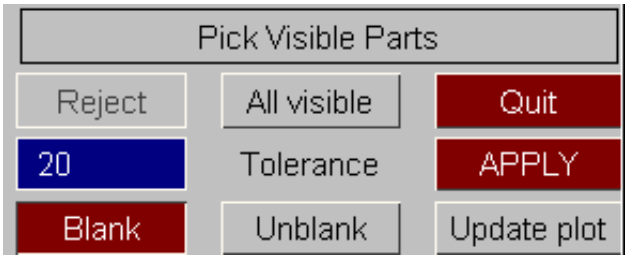
OFF Turns blanking **OFF** so that all items are drawn even if they have been blanked. This option does not reset the blanking status of items so that when blanking is turned **ON** again items that were previously blanked are still blanked.



These options can be used to modify the blanking status of a complete category of elements. In addition these option can also be used to **BLANK** / **REVERSE** / **UNBLANK** the whole model.

PICK	Pick items individually to be blanked / unblanked, (see below for more details)
AREA	Pick items by area to be blanked / unblanked (see below for more details)
KEY IN	Type in the ID of items to be blanked/unblanked
UPDATE PLOT	Redraw the image
STATUS	Lists the blanking status of all items
ATTACHED	With this option selected any item that shares a node with an item that is picked is also blanked or unblanked along with that item
UPDATE LEVEL	The **UPDATE LEVEL** controls whether items are blanked dynamically. If the **UPDATE LEVEL** is set to 3 then dynamic blanking is turned on.

6.1.1.1 **PICK** Pick items to be blanked / unblanked



If DYNAMIC blanking is turned on items are blanked as they are picked. When picking items the mouse buttons have the following function:

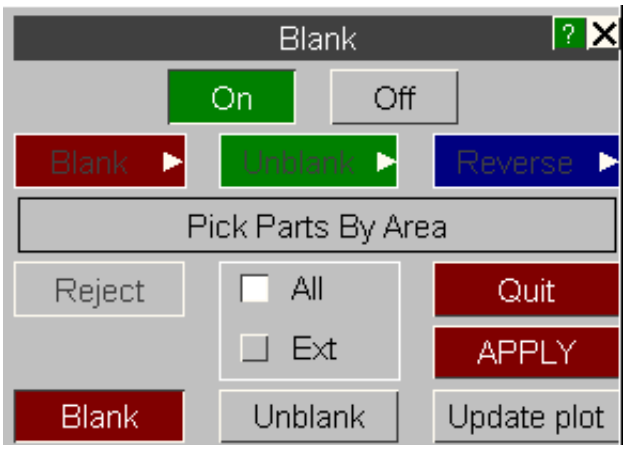
Left Mouse	Pick an item
Middle Mouse	Reject the last item selected (Update Level 1 & 2)
Right Mouse	Deselect an item (Update Level 1 & 2)

Other Options :

REJECT	Reject the last item selected (Update Level 1 & 2)
ALL VISIBLE	Select all items currently visible on the screen. Items outside the curtrent screen area are not selected.
QUIT	Quit without blanking/unblanking selected items (Update Level 1 & 2)
TOLERANCE	Define a screen tolerance for picking items.
BLANK	Items that are selected are blanked
UNBLANK	Items that are selected are unblanked
APPLY	Blank / Unblank selected items (Update Level 1 & 2) and then return to the main BLANK menu (Update Level 1,2 & 3)
UPDATE PLOT	Redraw the image with the currently selected items blanked / unblanked.

6.1.1.2 **AREA** Pick items by area to be blanked / unblanked

If DYNAMIC blanking is turned on items are blanked as they are picked. When picking items the mouse buttons have the following function.

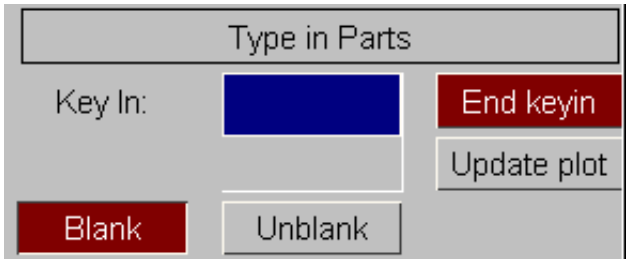


Left Mouse	Define a rectangle and select items within it.
Middle Mouse	Same as **Left Mouse**
Right Mouse	Define a rectangle and deselect items within it. (Update Level 1 & 2)

Other Options :

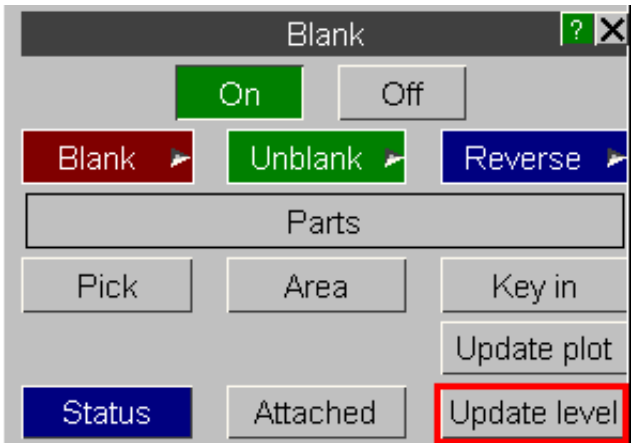
REJECT	Reject the first point selected
QUIT	Quit without blanking/unblanking selected items (Update Level 1 & 2)
BLANK	Items that are selected are blanked
UNBLANK	Items that are selected are unblanked
APPLY	Blank / Unblank selected items (Update Level 1 & 2) and then return to the main BLANK menu (Update Level 1,2 & 3)
UPDATE PLOT	Redraw the image with the currently selected items blanked / unblanked.

6.1.1.3 **KEY IN** Enter the ID of items to be blanked / unblanked



END SELECTION	Return to the main **BLANK** menu.
BLANK	Items that are selected are blanked
UNBLANK	Items that are selected are unblanked
APPLY	**Blank** / **Unblank** selected items (Update Level 1 & 2) and then return to the main **BLANK** menu (Update Level 1,2 & 3)
UPDATE PLOT	Redraw the image with the currently selected items blanked / unblanked

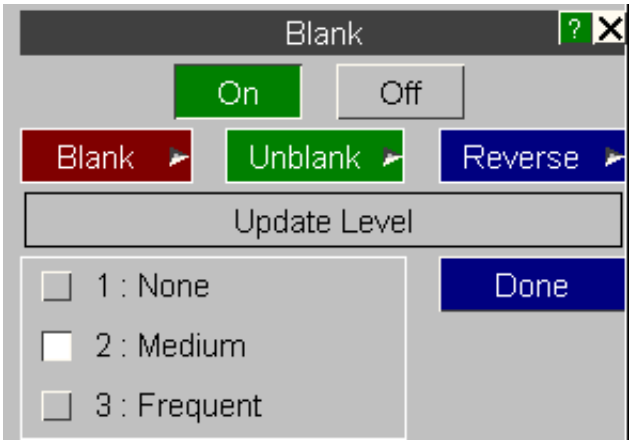
6.1.1.4 **UPDATE LEVEL** Image update frequency



The top half of the menu provides ways to select the items to be blanked and unblanked while the bottom half is used to choose the type of item that is going to be blanked / unblanked.

1 (NONE) The image is never updated automatically to show the effect of blanking. You must redraw the image (eg DR, SH, etc) to see the effect of changes. This is not recommended unless you have a very slow connection to your display.

2 (MEDIUM) Default behaviour. (Un-)Blanking something does not cause the display to update, but any subsequent viewing operation (eg zoom, dynamic view, etc) will result in the image being updated to show the effect of changes.



3 (FREQUENT) Immediate update. Every time you (un-)blank something the display will be redrawn immediately to show the effect of the change. Very large models on slow displays may become cumbersome if this is used.

It is recommended that you keep the default **UPDATE LEVEL** of **2**, and use the **UPDATE PLOT** button explicitly to see the effect of changes. (The "Quick Pick" blanking option provides "instant" blanking, and is a better way of achieving this behaviour.)

6.1.2 Entity Types

The bottom section lists the generic entity types that the model contains and this section controls the type of items that are selected by the **PICK** and **AREA** and **KEY IN** options.

By default this section shows a list of all the PARTs that the model contains. The blanking status of an item can be changed by clicking on its entry in the list. The list of items is colour coded as follows :

RED The whole of the item is blanked.

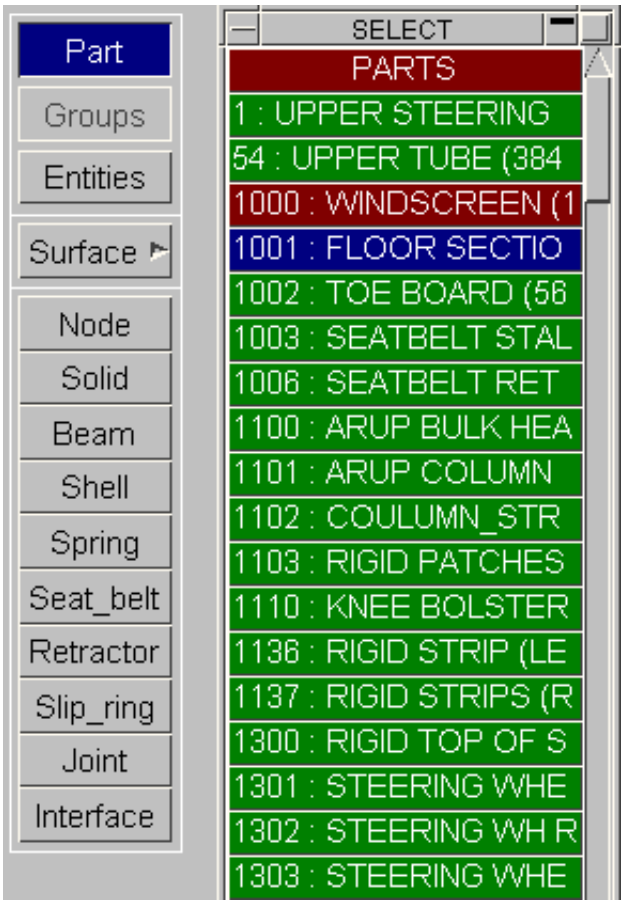
GREEN The whole of the item is unblanked.

BLUE Some of the item is blanked.

If a BLUE menu entry is clicked on the item will be completely blanked. Clicking on the menu entry a second time will then completely unblank the item - **it is not possible to return to the partially blanked state.**

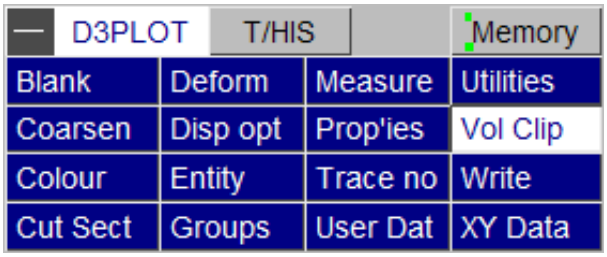
If an option other than PARTS is selected then the list is automatically updated to list the appropriate items. To reduce the number of generic entity types the **SEATBELT** and **SURFACE** buttons are linked to popups containing related entity types.

If DYNAMIC blanking is active then items are blanked / unblanked as they are selected in the list. If DYNAMIC blanking is not active the menu will be updated as items are selected but the image will not be updated until either the view is changed (rotated, zoomed etc) or the image is explicitly redrawn (**HI**, **CT**, **SH** etc).



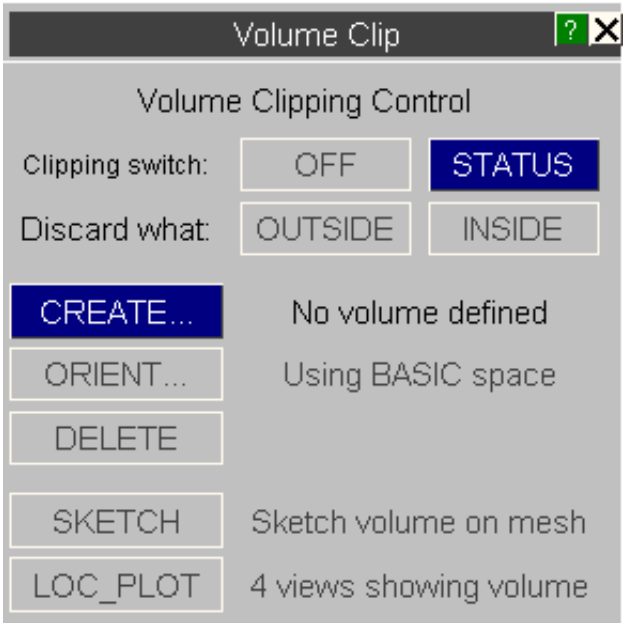
Next section

6.2 VOLUME_CLIPPING



By default no volume clipping or cut-sections are in effect, and pressing the **VOLUME CLIPPING** button will give the main panel in its basic state, as shown in the figure (right).

Since in the initial volume-clipping state nothing is active only the **CREATE** function, as shown here, is available. Other options are greyed out.



Volume Clipping is a "per window" attribute.

Volume clipping definitions apply to all those windows which have their **`W1 .. Wn`** tabs set. Clipping takes place in the specified space system in each window, and will apply to all models in that window.

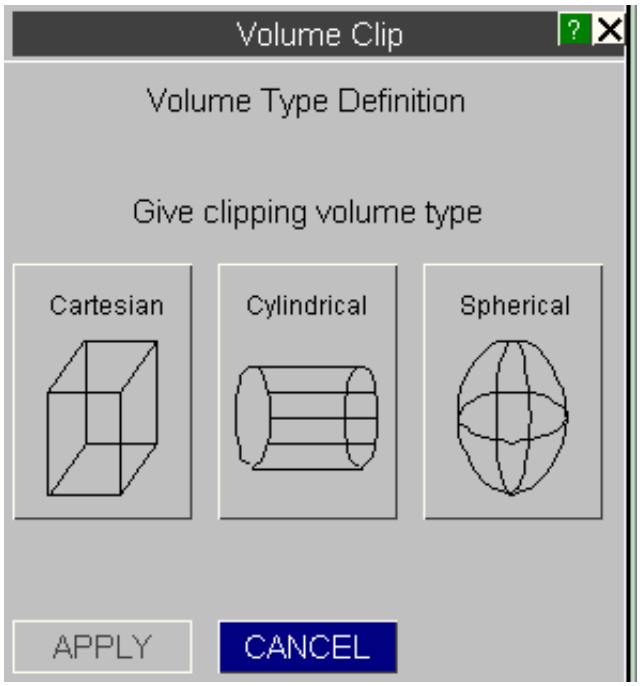
If you use any of the "pick node" options to obtain coordinates you will have to specify which model you are picking from, but thereafter the coordinate is stored independently of its source model (ie simply as a number) and can apply to any models.

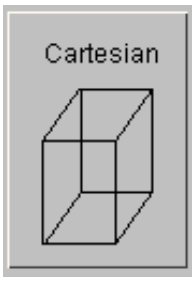
6.2.1 **CREATE** Creating a new clipping volume

When creating a volume you must first define its type.

The options are **Cartesian**, **Cylindrical** and **Spherical** as shown in the figure (right).

Pick a type, then hit **APPLY** to continue.

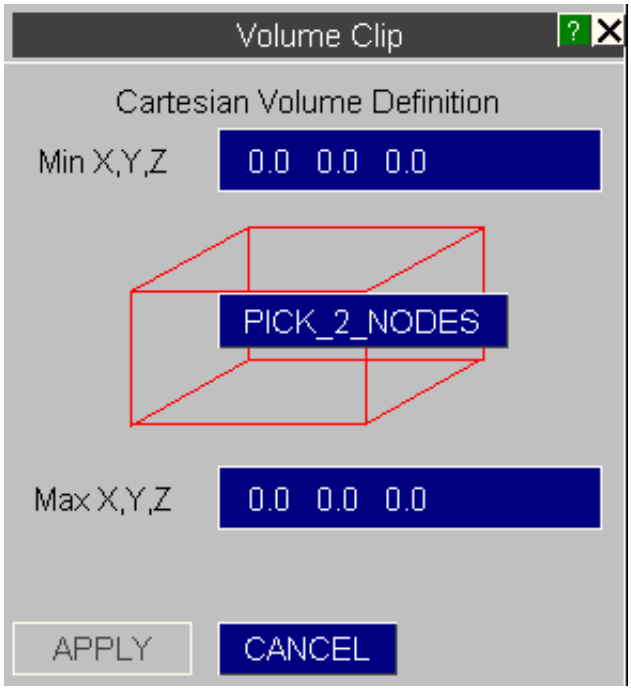


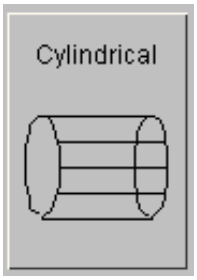


Defining a **Cartesian** volume

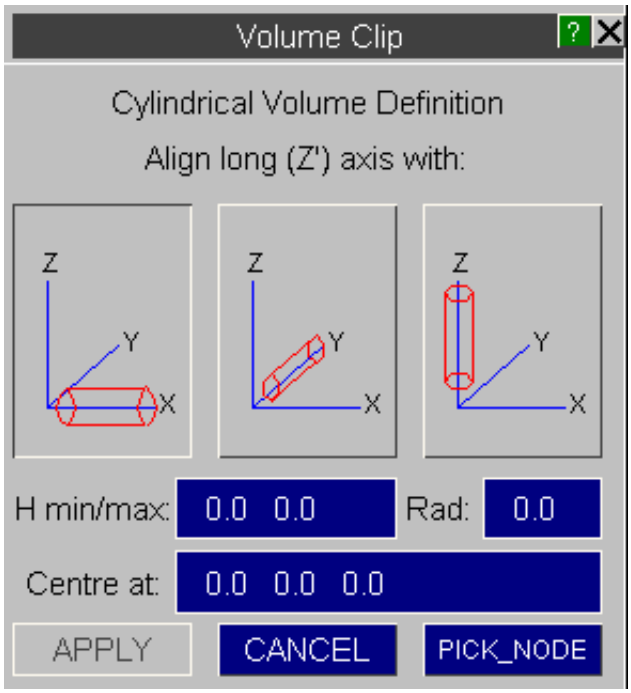
A cartesian volume is defined by:

Corners: Type in the coordinates of two diagonally opposite corners at **Max/Min X**,**Y**,**Z**, or use **PICK_2_NODES** to locate corners at nodal coordinates.





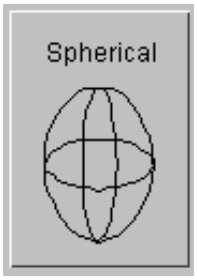
Defining a **Cylindrical** volume



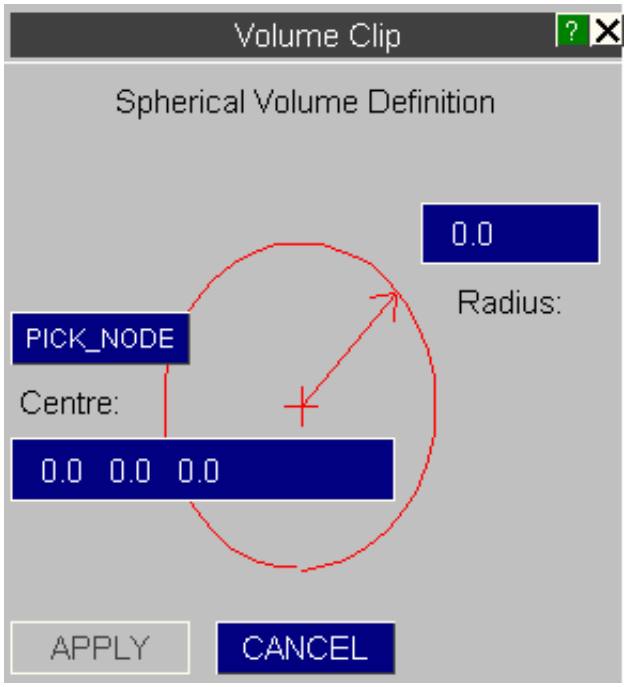
A cylindrical volume is defined by:

Origin:	Type in at **Centre at**, or use **PICK_ NODE** to locate at a nodal coord.
Radius:	Type in at **Rad:**
Height:	Type in the maximum and minimum height at **H min/max**:

Align the long (height) axis on one of the global X, Y or Z axes with the relevant button.



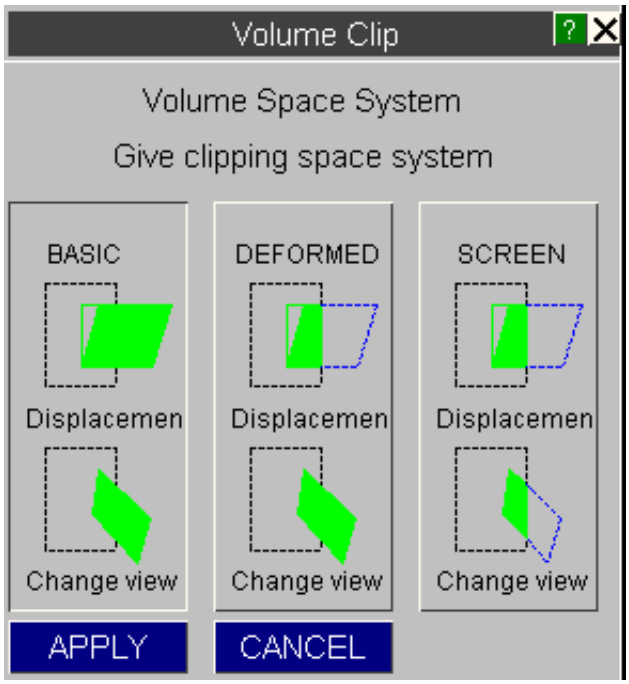
Defining a **Spherical** volume



A spherical volume is defined by:

Centre:	Type in at **Centre**, or use **PICK_NODE** to locate it at a nodal coordinate.
Radius:	Type in at **Radius:**

6.2.2 **ORIENT** Defining a space system for volume clipping

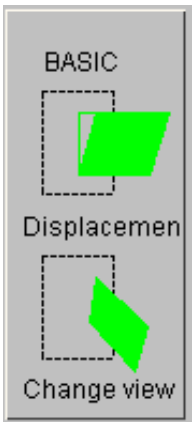


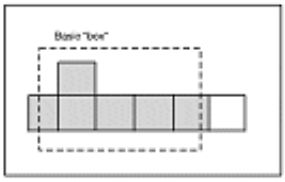
When you **CREATE** a volume, or **ORIENT** an existing one, you must define a space system for it. This figure shows the space system definition panel.

There are three options: **BASIC**, **DEFORMED** and **SCREEN** space.

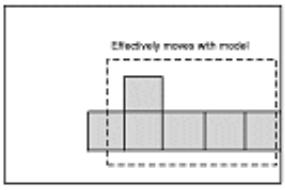
These have the following meanings:

6.2.2.1 **BASIC**

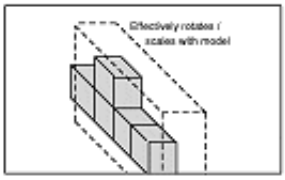




BASIC space volume: Shaded elements are drawn. (Element centres inside box)



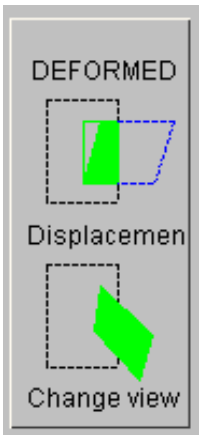
Effect of deformation: Same elements remain visible. (ie the "box" moves with the elements.)

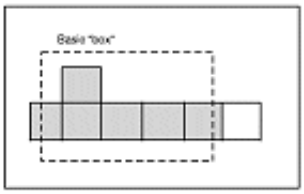


Effect of rotation/scaling: Same elements remain visible.

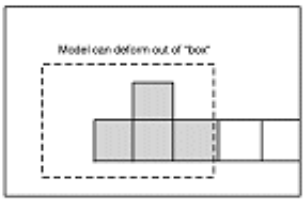
Clipping is based on undeformed nodal geometry. So the same elements are always visible regardless of their deformations or any changes of view.

6.2.2.2 DEFORMED

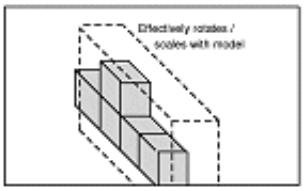




DEFORMED space volume: Shaded elements are drawn. (Element centres inside box)



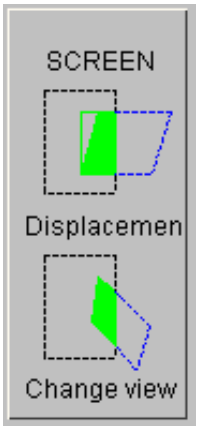
Effect of deformation: Element visibility changes as elements pass in & out of the box.

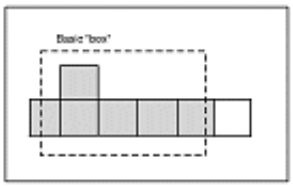


Effect of rotation/scaling: Same elements remain visible.

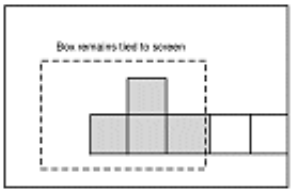
Clipping is based on the deformed nodal geometry at each state. So elements may pass in and out of the clipping volume as they move and deform.

6.2.2.3 SCREEN

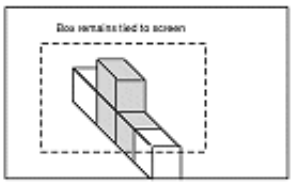




SCREEN space volume: Shaded elements are drawn. (Element centres inside box)



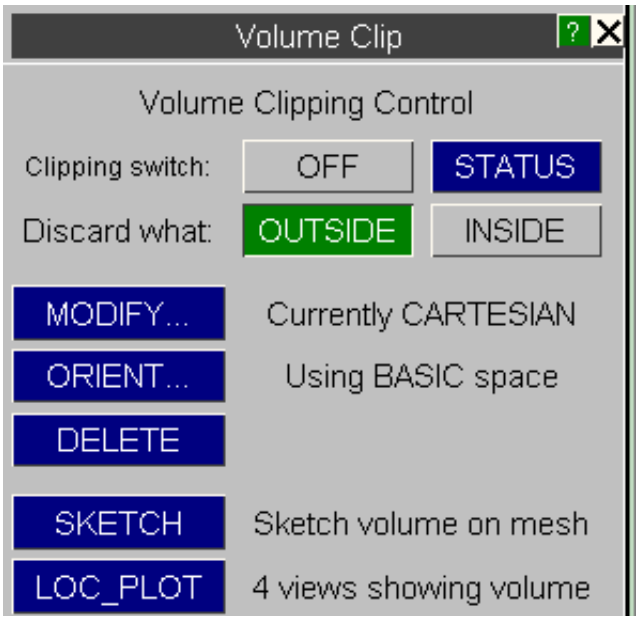
Effect of deformation: Element visibility changes as elements pass in & out of the box.



Effect of rotation/scaling: Elements may move in & out of clipping "box".

Clipping is tied to screen coordinate space. Thus rotation and scaling operations, as well as deformations, may move elements in and out of the volume

6.2.3 Actions once a volume has been created.



Once you have created a clipping volume then the other options on the master panel become "live". These are described below:

6.2.3.1 **Clipping switch** Turning volume clipping on/off

By default volume clipping is **OFF**, even when you have just created a volume. You must turn it **ON** for clipping to take effect.

6.2.3.2 **Discard what** Discarding entities inside or outside the volume

By default entities **OUTSIDE** the volume are discarded, but you can invert the effect so that everything **INSIDE** the volume is discarded instead.

6.2.3.3 **MODIFY...** Modifying the existing volume

When a volume already exists you can **MODIFY** it. This takes you through the volume definition sequence described in Section 6.2.1 for creating a new volume. (Note that this button is called **CREATE** if no volume exists.)

6.2.3.4 **ORIENT...**Change the space system of an existing volume

You can modify the space system (**Basic**, **Deformed**, **Screen**) of an existing volume, without changing its geometry. This takes you through the space definition sequence described in Section 6.2.2.

6.2.3.5 **DELETE** Deletes the existing volume definition completely

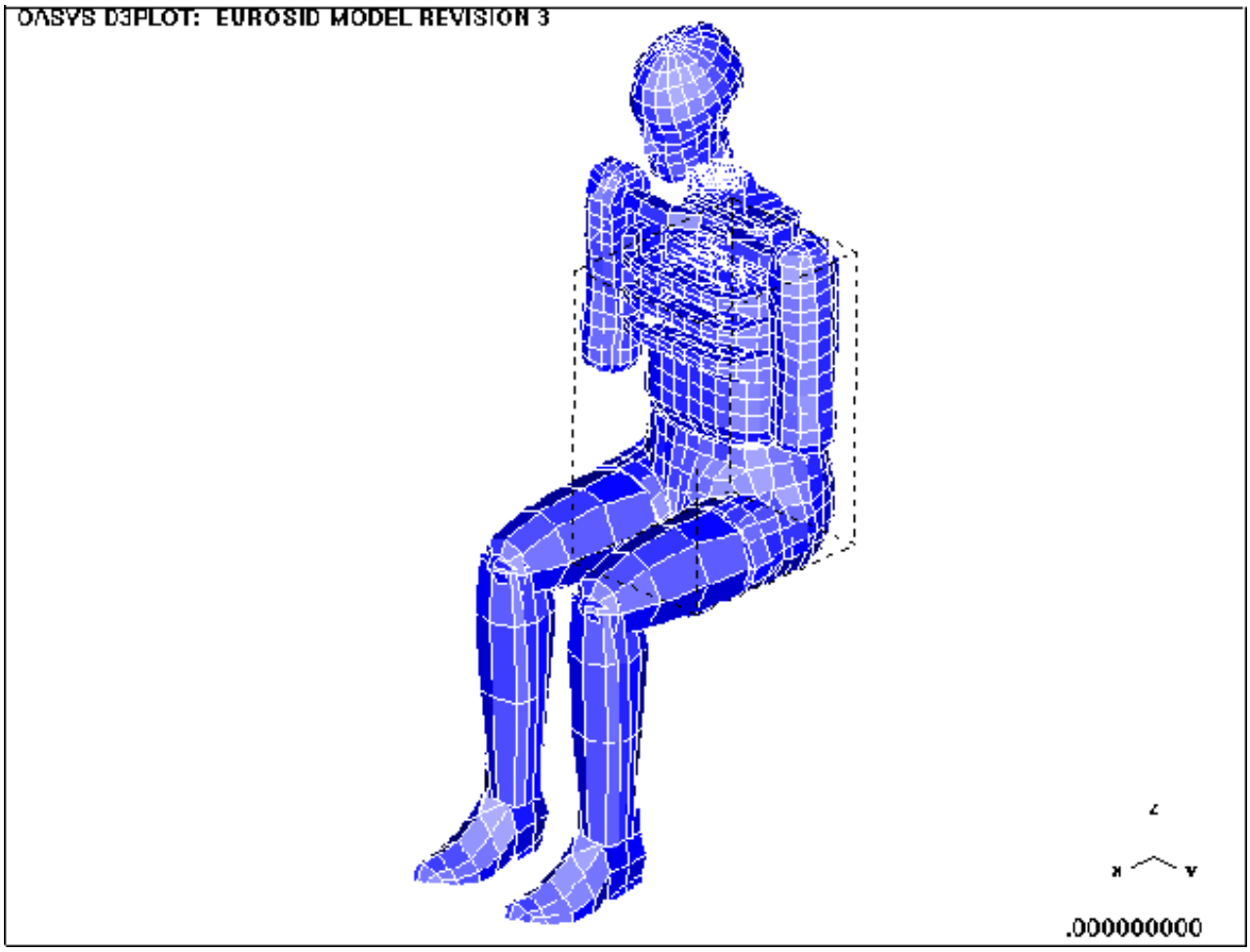
You can **DELETE** the existing volume completely. This returns the clipping control panel to its initial state as shown in the figure "Volume-clipping panel with no volume current". (To suppress, but not delete it, switch it **OFF**.)

6.2.3.6 **SKETCH** Sketch the volume on the current geometry

This figure shows a typical model with the current (Cartesian) clipping volume **SKETCH**ed on it.

The outline of the current volume is drawn in dashed outline on the current screen image.

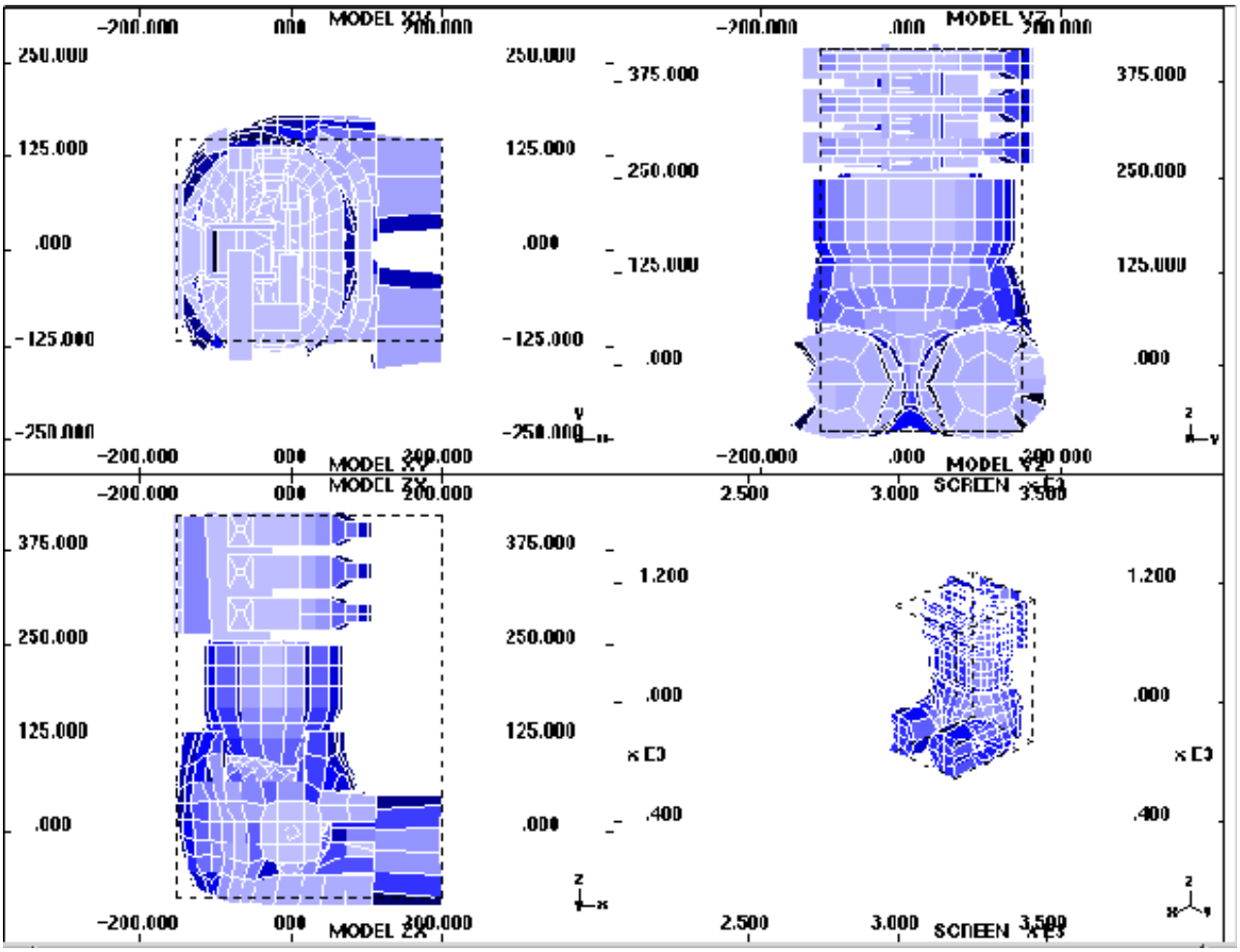
In this example the clipping switch has been turned **OFF**, so nothing is removed by clipping.



6.2.3.7 **LOC_PLOT** “Location” plots showing 4 views of the current volume

This figure shows the image above, with the volume-clipping switch turned on, drawn as a "location" plot. The display mode used is that of the most recently issued drawing command.

This draws 3 standard views (on **XY**, **YZ** and **XZ**), and also the current view in the bottom right quadrant. The **GRATICULE** (see **DISPLAY_OPTIONS**) is also turned on to give you numeric feedback.



Note: The **SKETCH**ed outline of the volume, and all graphics in location plots, are treated as transient graphics, and are **not** saved to a laser file if laser output is turned on.

6.2.3.8 Further notes on volume clipping

Note 1: Clipping is calculated using the simple test: "is the element centre within the current volume?". Then either the whole element is displayed, or it is not drawn at all. No interpolation across elements takes place.

This can give an effect rather like taking bricks out of a wall as shown in this figure.

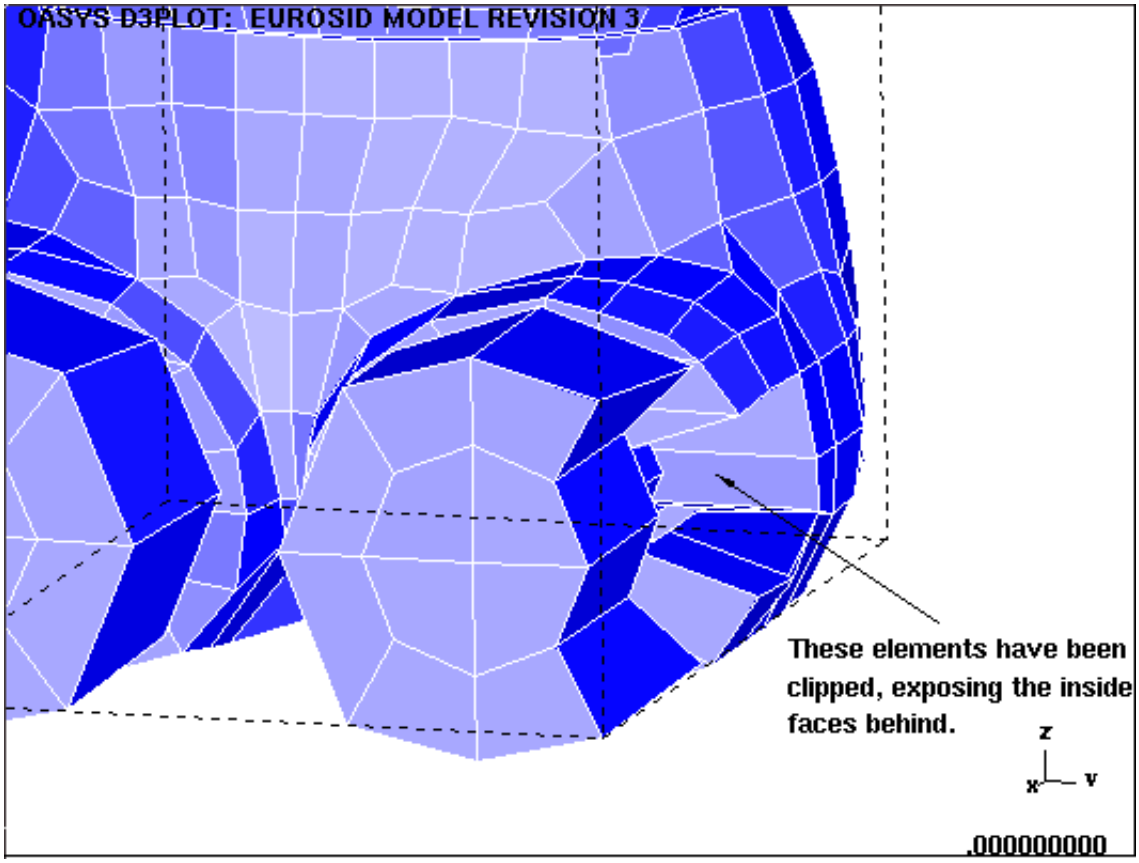
Note 2: Volume clipping does not work with stonewalls. This is because their geometry is at best strange, and often has infinite dimensions, making it too hard to implement.

Note 3: On 3D devices the graphics mode will be switched temporarily back to 2D when creating "location" plots.

Note 4: You cannot screen-pick entities from "location" plots.

Note 5: When a clipping volume exists you will find that the CV (Current Volume) button is live in entity <list> entry panels. (See Section 6.0.2, and its accompanying figure .) This provides the option of selecting entities within the current volume.

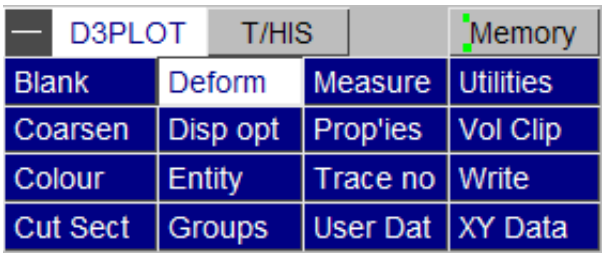
Note 6: Volume clipping can affect other parts of D3PLOT, as shown in the following table:



Function affected by clipping	Relevant D3PLOT command	Section
Averaging of element data at nodes	**AVERAGING...** **Clipping ignored switch**	4.2.4.6
Calculation of free edges	**DISPLAY_OPTIONS** **FREE_EDGES...** **CLIPPING -> EDGES**	9.11
Exclusion of elements from a "scan" for maxima/minima	**WRITE** **SCAN**	6.7.2

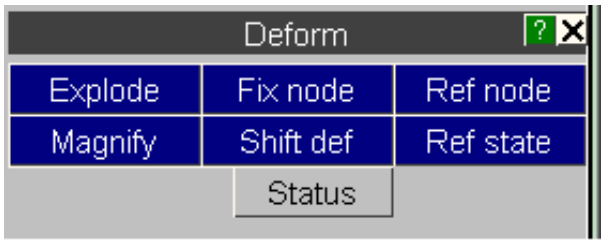
Next Section

6.3 DEFORM Deforming geometry.



The **DEFORM** command contains four functions which "deform" geometry in various ways: exploding parts, magnifying displacements, fixing a node in space and fixing the model in space.

This figure shows the generic **DEFORM** panel, which gives access to its functions.



EXPLODE_PARTS	Artificially separates parts by applying "explosion" vectors to them
MAGNIFY_DISPLACEMENTS	Allows scales other than 1.0 to be applied to graphical displacements.
FIX_NODE	Subtracts the displacement at a node from that at all others, effectively "fixing" it in model space.
SHIFT_DEFORMED	Fixes three nodes, forming a local coordinate system, against which all displacements are drawn.
REFERENCE_NODE(s)	Makes results relative to those at one or three nodes
REFERENCE STATE/MODEL	Makes results relative to a "reference" state in this or another model

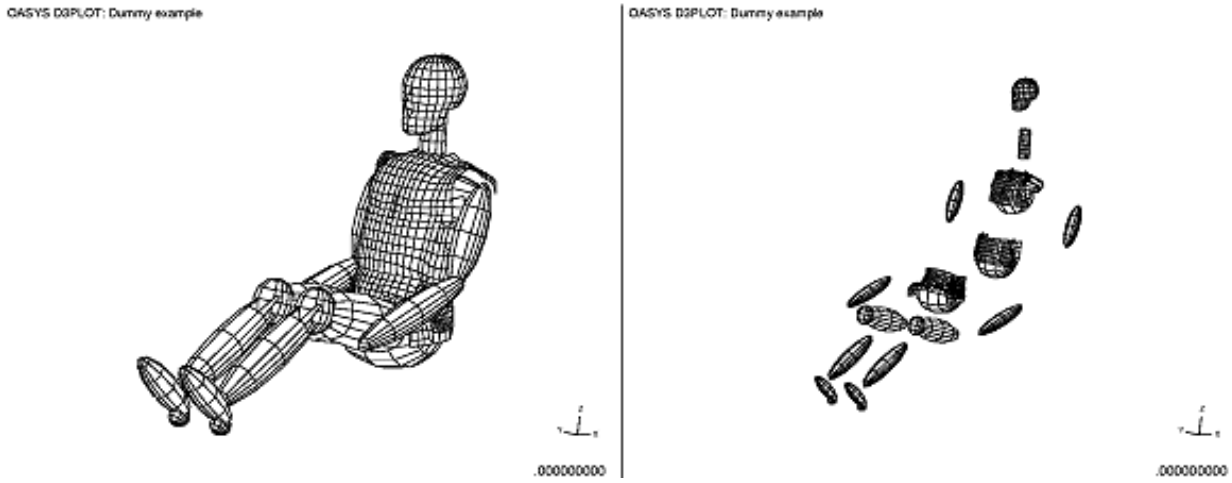
DEFORM options apply at a mixture of "per window" and "per model" levels.

All the options will apply to the windows selected by the **`W1 .. Wn`** tabs, but wherever node labels are used the following rules apply:

- Node labels will be mapped onto all relevant models.
- If a node does not exist in a particular model then that feature will be disabled in that model.
- If you screen-pick nodes you have to say which model they are to be picked from, but once picked the "label" rules above apply.

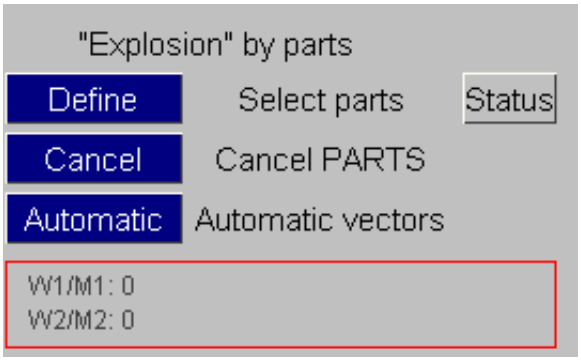
6.3.1 **EXPLODE_PARTS** Separating (“exploding”) parts

"Exploding" a part is done by applying a [dx,dy,dz] vector to all nodes of that part, which has the effect of moving it bodily to a new location. By default no explosion vectors are set, but you can define, modify and cancel vectors for any part(s) at will. The figure below left shows an unexploded, and below right, an exploded dummy model:



This figure shows the basic "Explosion" control panel in its initial state.

In this example no explosion vectors have been defined yet, as can be seen in the status feedback area.

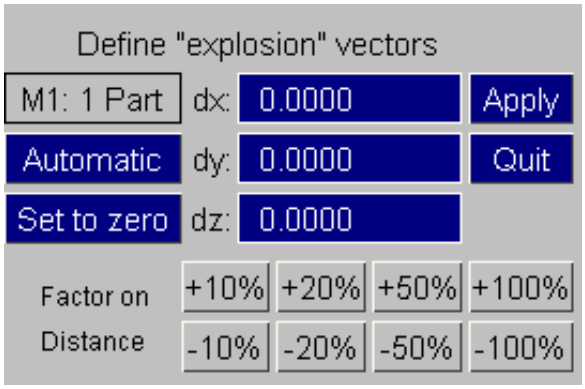


6.3.1.1 **DEFINE** Defining explicit explosion vectors for a <list> of parts.

If you know exactly which parts you want to "explode", and by how much, use **DEFINE**. This requires you to define a <list> of parts, using the standard selection panel, then for each part it gives you the explosion vector definition panel shown in the following figure:

For each part you can define:

dx,**dy**,**dz**	Explicit vectors;
AUTOMATIC	Let D3PLOT calculate vectors (based on vector from model C.of.G to part C.of.G).
SET TO ZERO	Reset all 3 vectors to zero.
Factors...	Provides a simple way to factor the current vectors by known %age amounts.



You can use the definition methods above in any order: for example use **AUTOMATIC** to get an initial estimate, then **Factor** them, or modify them by hand. The vectors are only stored when you give the **DONE** command. The vectors you define will take effect the next time you issue a plotting command.

6.3.1.2 **CANCEL** Cancelling (resetting to zero) explosion vectors

Explosion vectors remain in force until you change or **CANCEL** them explicitly. Use **CANCEL**, then select a <list> of parts to have their vectors zeroed. The effect will be seen the next time you issue a plotting command.

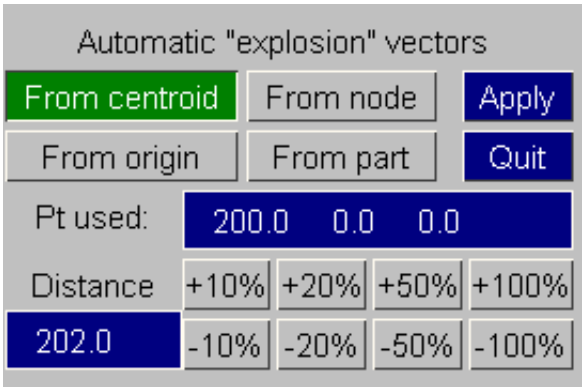
6.3.1.3 **AUTOMATIC** Automatically generated vectors for a <list> of parts

Typing in vectors for a long list of parts can get tedious, so it is possible to get D3PLOT to generate vectors for you automatically. These are based on the vector from a defined position (by default the centre of the model) to the centre of gravity (C.of.G) of each part, multiplied by a known factor.

This figure shows the **AUTOMATIC** vector definition panel. You define the centre from which vectors are calculated from one of:

CENTROID	Model centre of gravity
NODE	Nodal coordinate
ORIGIN	Coordinate [0,0,0]
MATL	C.of.G of a part
Pt used	Type in a coordinate

Then define a (non-zero!) **Distance**, or apply a %age factor to the existing value.



When you have defined the centre of explosion correctly press **APPLY**, and you will be asked to define a <list> of parts to which to apply vectors. Vectors for each part will be calculated based on the distance from the part C.of.G to the centre defined here, factored in proportion to the **Distance** value. The vectors generated are not usually ideal, but they provide a good starting point from which they can be "tweaked" to give the required image.

6.3.1.4 Notes on explosion vectors.

Note 1: Vectors only affect plots, they have no influence on nodal coordinates used for X-Y plotting, or in **WRITE**, or upon the calculation of element volume etc.

Note 2: Explosions are applied as vectors added to nodal coordinates. Where two parts share a common node there is an ambiguity: should vectors be cumulative or, if not, which value whould be used? In this case D3PLOT uses the vector of the lowest numbered part, and ignores the remainder. So try to avoid this situation or, if you cannot, be prepared for elements on the border of two parts with dissimilar vectors to appear to be stretched.

Note 3: Explosion can be used in conjunction with the other options in the **DEFORM** menu. Transformations to nodal coordinates are applied in the order:

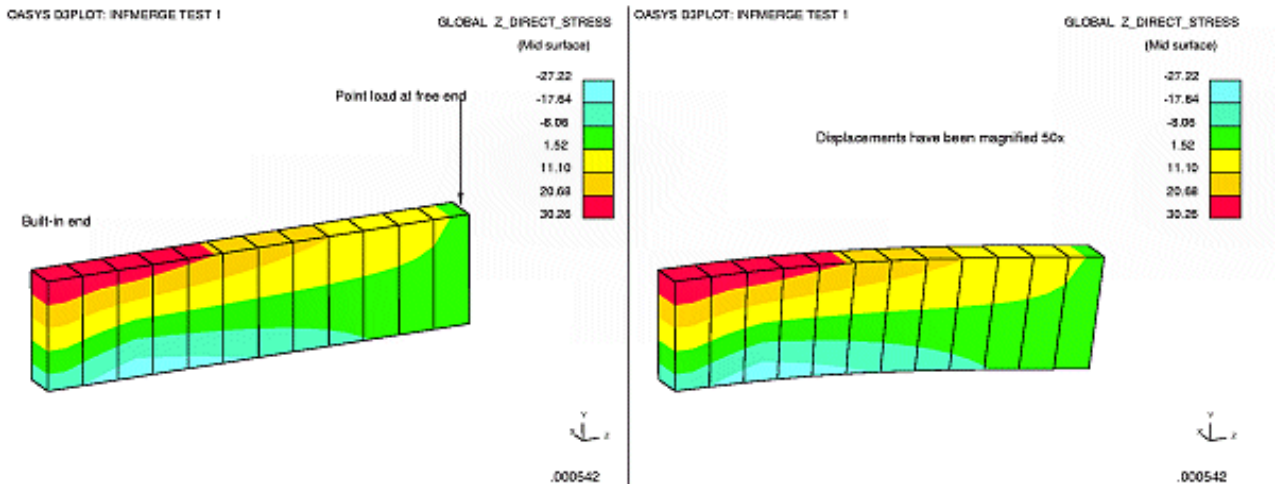
FIX_NODE or **SHIFT_DEFORMED** (Mutually exclusive)

MAGNIFY_DISPLACEMENTS

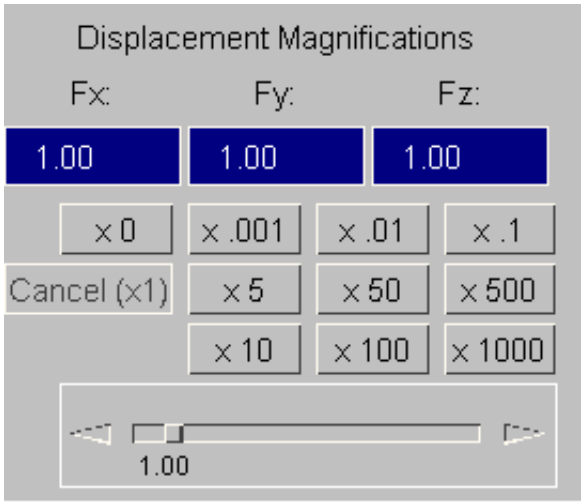
EXPLODE

6.3.2 **MAGNIFY_DISPLACEMENTS** Factoring nodal displacements

By default D3PLOT plots nodal coordinates at their true positions: a displacement factor of 1.0. However there are times when you may wish to factor displacements, for example when stresses are in the linear elastic range and displacements hardly visible. The figure below left shows an example of unfactored, and below right, factored (magnified by 50), displacements for a cantilever subject to a point load on its end.



The **MAGNIFY_DISPLACEMENTS** panel is shown in the figure (right).
To enter factors type in [**Fx,Fy,Fz**], or use one of the pre-programmed factors (**x5**, **x10**, etc). The **CANCEL** button sets all factors back to the default of 1.0. The **Factor on Curr** slider applies the given factor to the current values - an easy way of setting any value.



When you have defined the factors use **DONE** to return and apply them.

Factors take effect the next time you update the plot, and stay in effect until changed again.

Note 1: Factors only apply to plots. They have no effect on written of X-Y data output, calculation of volumes etc, or contoured values of displacement.

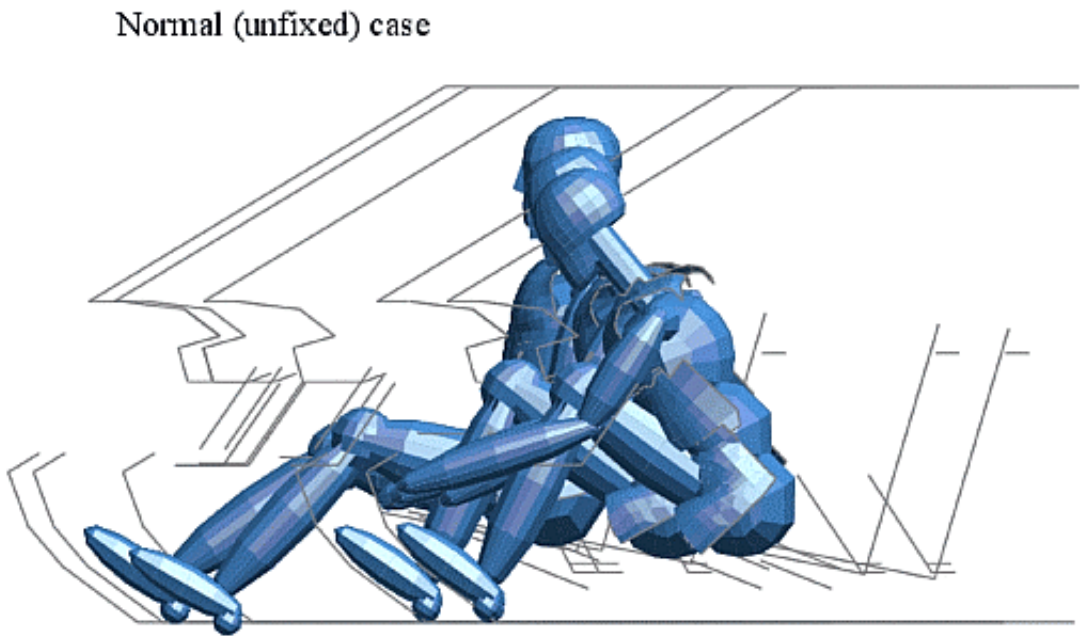
Note 2: Magnified displacements may be used in conjunction with the other **DEFORM** options: see Note 3 in Section 6.3.1.4 for the order of operations.

6.3.3 **FIX_NODE** Fixing a node position despite displacements

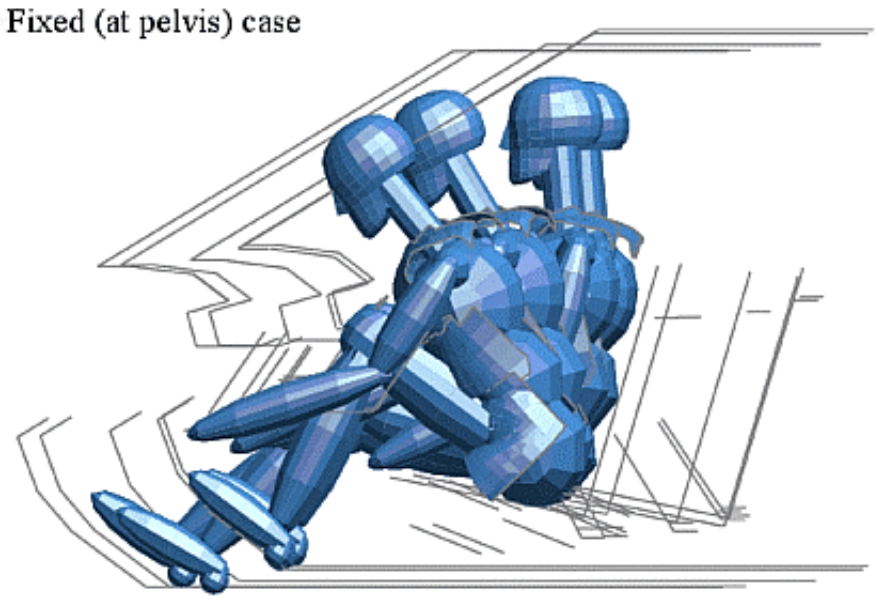
In some circumstances a model may move a long way between successive states, and it can be inconvenient to have it progressively disappearing off the screen.

This figure shows a few frames during the assembly of an animation of a dummy sled test analysis.

In these tests the sled is pulled backwards to mimic the deceleration during a crash, and it moves off the screen as a consequence. So a simple translation to bring it back to the undeformed position will suffice.



This second figure shows what happens when a node in the dummy's pelvis is fixed using **FIX_NODE.**



The **FIX_NODE** facility allows you to specify a node that remains fixed at its undeformed position, regardless of any displacements that may occur. This is implemented by finding the displacement vector of the node at each complete state, and subtracting that vector from the coordinates of every node in the model. In the example above one would fix a node on the sled, which would then appear to be fixed in space, and simply see dummy motion within it.

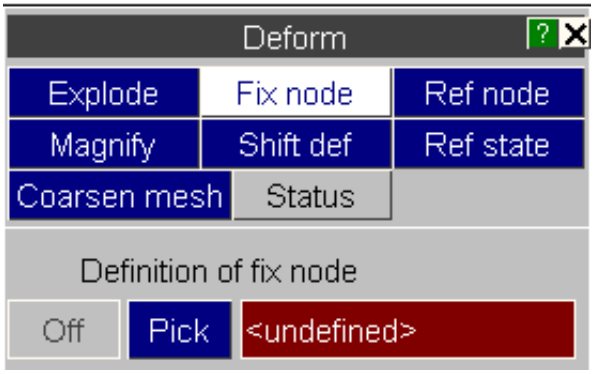
(If a rotation as well as a translation is required you can use **SHIFT_DEFORMED** instead: see Section 6.3.4.)

This figure shows the **FIX_NODE** control panel in its default state: no node is fixed.

To fix a node **PICK** it, or type in its label. Once defined this mode can be switched on or off.

FIX_NODE applies a model space offset to what is drawn: it is a translation of the model, updated for each state.

(It is not the same as **VIEW, CN** (Centre on node) which is a purely graphical transformation that sets the viewing centre for rotations.)



Note 1: **FIX_NODE** (which applies a translation) and **SHIFT_DEFORMED** (which applies both a rotation and a translation) are mutually exclusive: you can only have one or the other active at one time.

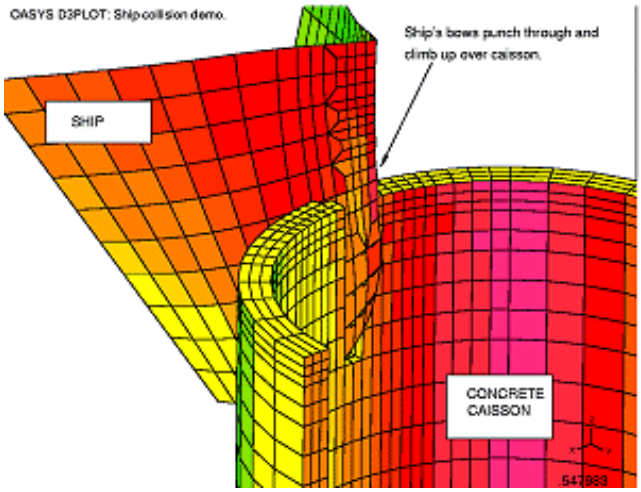
Note 2: The node used in **FIX_NODE** (which affects the graphical displacements) is the same as that used as a single **REFERENCE_NODE** (which affects the contoured and reported values). They may be used separately or together.

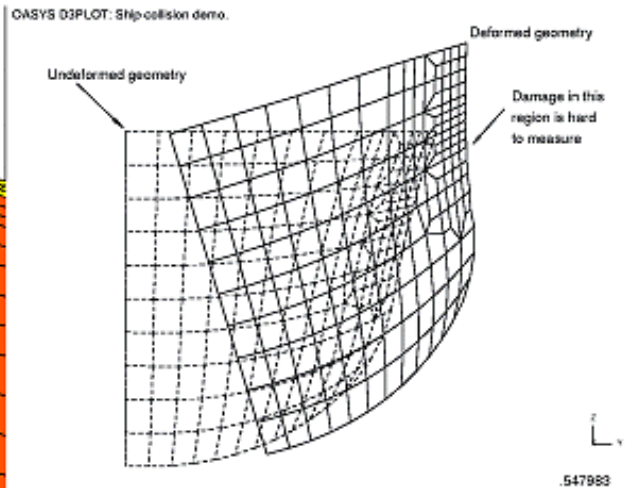
Note 3: **FIX_NODE** can be used at the same time as "explosion" vectors and magnified displacements: see Note 3 in Section 6.3.1.4 for the order of application.

6.3.4 **SHIFT_DEFORMED** Translating and rotating a model back to its undeformed position.

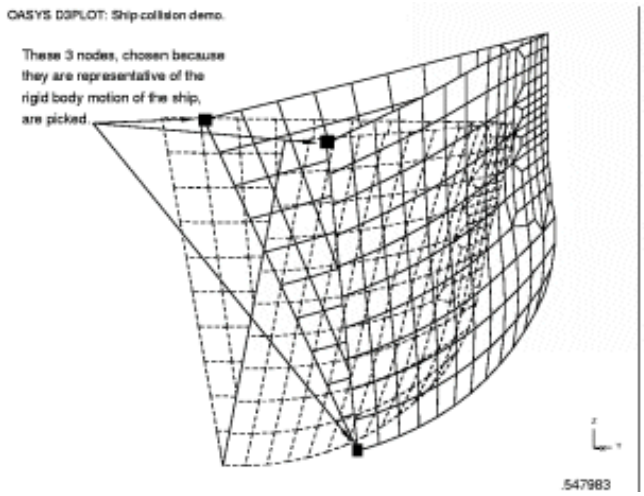
Sometimes it is useful to be able to move a deformed structure back to its undeformed position, for example to measure knock-back (crush) following an impact. In many cases this will involve applying a rotation as well as a translation, and the **FIX_NODE** option described in Section 6.3.3 (which only applies a translation) will not be adequate.

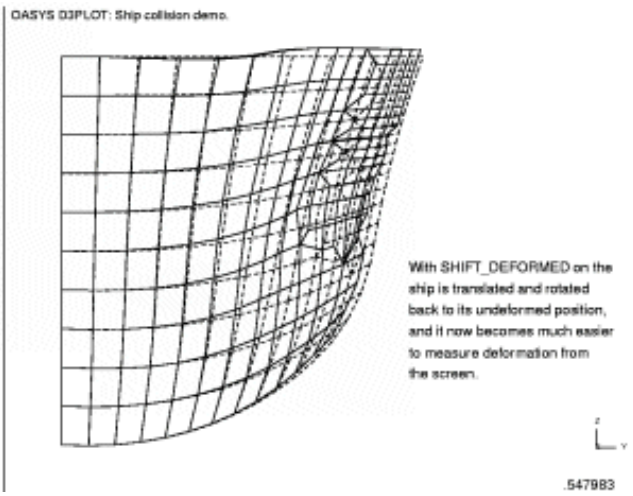
Consider the following example: a ship hits a concrete caisson, punches a hole through it with some damage to its bow plates, and also pitches up as at tries to climb over the caisson. Measure the damage to the bow plates. The situation is shown in the figure below left, and in the figure below right the deformed and undeformed shapes of the ship are shown. Clearly the rotation the ship has undergone makes it hard to measure the deformation





By using **SHIFT_DEFORMED** to pick three nodes that are representative of the rigid body motion of the ship, translation and rotation can be applied to bring the deformed geometry back to overlay the undeformed, making measurement possible: see the figures left and right below).

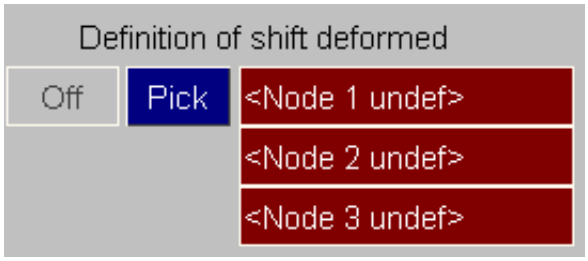

OASYS D3PLOT: Ship collision demo.
These 3 nodes, chosen because they are representative of the rigid body motion of the ship, are picked.
.547983


OASYS D3PLOT: Ship collision demo.
With SHIFT_DEFORMED on the ship is translated and rotated back to its undeformed position, and it now becomes much easier to measure deformation from the screen.
.547983

The **SHIFT_DEFORMED** panel is shown right.

You can screen-**PICK** the nodes, or type them in directly.

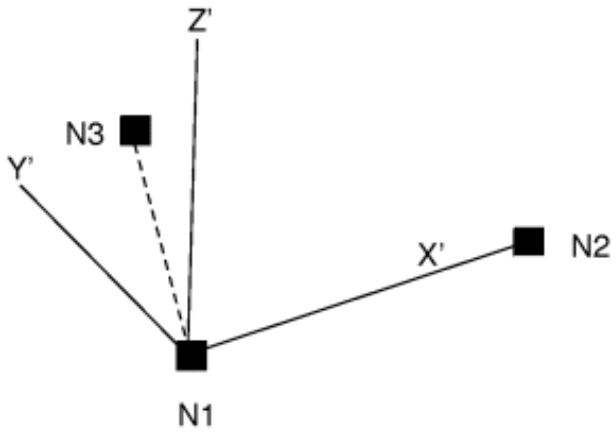
Once defined this mode can be switched on or off at will.



Choosing sensible nodes for SHIFT_DEFORMED.

The three nodes you choose form a right handed coordinate system, so they must not be colinear (or become colinear due to displacements), and the order of their definition is significant: see the figure below.

The three nodes for SHIFT_DEFORMED form a right-handed coordinate system.



Vector N1N2 defines the local X' axis
N1N2 x N1N3 defines the local Z' axis
The Y' axis is obtained from Z' x X'

The (-ve) displacement of N1 is applied to the whole model.

The local coordinate system [X',Y',Z'] is defined as shown here, and the inverse of this is applied to rotate the model

back to its undeformed state.

The displacement of node 1 is subtracted from all nodes in the model to bring it back to the undeformed position.

(**FIX_NODE** applies this translation only, thus it is a subset of **SHIFT_DEFORMED**, which is why the two operations cannot be used at the same time.) You should try to choose three nodes whose relative position will not change too much as the model deforms, so that their motion is representative of the rigid body motion of the structure as a whole. And node 1, from which the rigid body translation if computed, is the most significant. For example in a frontal impact car crash analysis you should nodes at the back of the car. If your model has some rigid bodies then nodes on them would be ideal.

Note 1: **SHIFT_DEFORMED** cannot be used at the same time as **FIX_NODE**, since the translations they apply would conflict.

Note 2: **SHIFT_DEFORMED** can be used in conjunction with "explosion" vectors and magnified displacements. The order of application is given in Note 3, Section 4.8.14.

Note 3: **SHIFT_DEFORMED** uses the same three nodes as those in **REFERENCE_GEOMETRY**. The difference is that shifting the model simply changes the graphics displayed, whereas reference geometry changes the data values contoured and output. They can be used in conjunction or separately.

6.3.5 **REFERENCE_NODES** Calculating results with respect to one or three nodes.

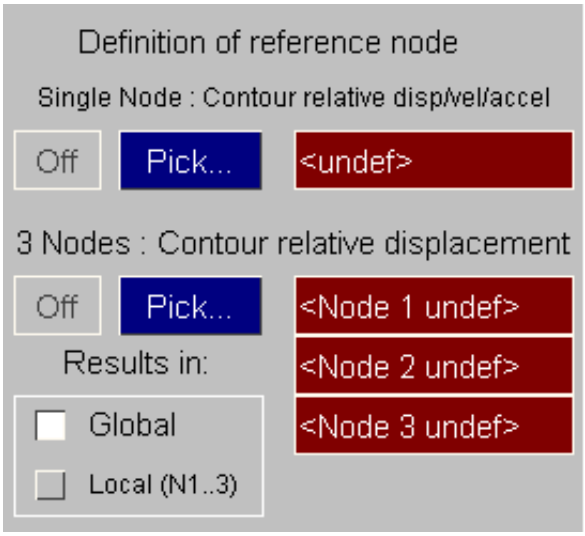
FIX_NODE and **SHIFT_DEFORMED** above affect only how the current image is displayed, they do not change the computed values which are contoured or reported.

REFERENCE_NODES, on the other hand, does not affect the display at all, rather it modified the values that are computed to make them relative to those at the nodes chosen. This feature allows intrusion or relative deformation to be contoured. Two mutually exclusive options are available:

- **Single node**: Displacement, Velocity and Acceleration values are reported relative to that node.
- **Three nodes**: Displacement only is reported relative to node 1, in the coordinate system formed by N1N2N3.

  In the Three nodes case results can be reported in either the global or the local (N1N2N3) coordinate system.

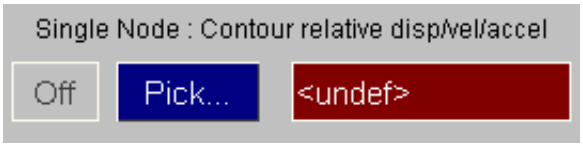
The "**Single**" and "**Three**" node cases are mutually exclusive, you cannot have both active at one time.



6.3.5.1 Defining one or three nodes

Single Node

You pick a single node <N_0>.



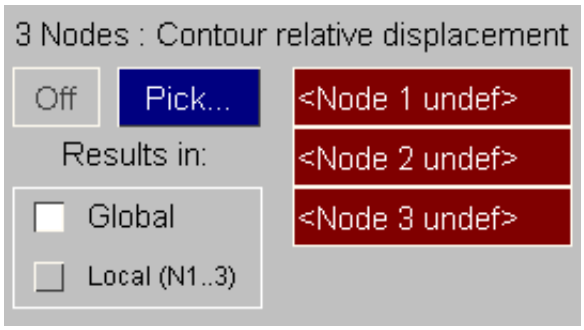
Displacements, Velocities and Accelerations are calculated with respect to the value at that node. For example if **V** is a velocity vector:

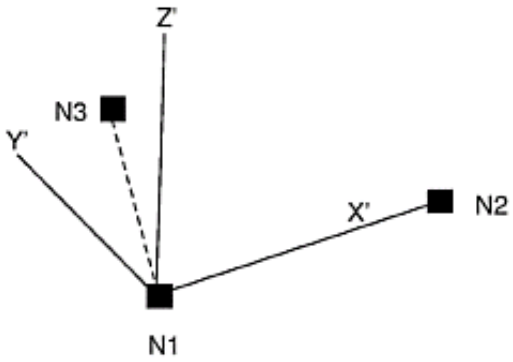
$$V'_N = V_N - V_0$$

V'_N = modified velocity vector at node <n>
V_N = original velocity vector at node <n>
V_0 = current velocity vector at reference node <N_0>.

Three Nodes

You pick three nodes <N_1,N_2,N_3>. N_1 is the origin, and the nodes form a right-handed coordinate as for SHIFT_DEFORMED above.





Displacements (only) are calculated with respect to this system such that for displacement vector D:

$$D'_N = R \cdot [D_N - D_O]$$

D'_N = modified displacement vector at node <n>
D_N = original displacement vector at node <n>
D_0 = current displacement vector at reference node <N_0>
R = the rotation matrix to transform back to the selected coordinate system

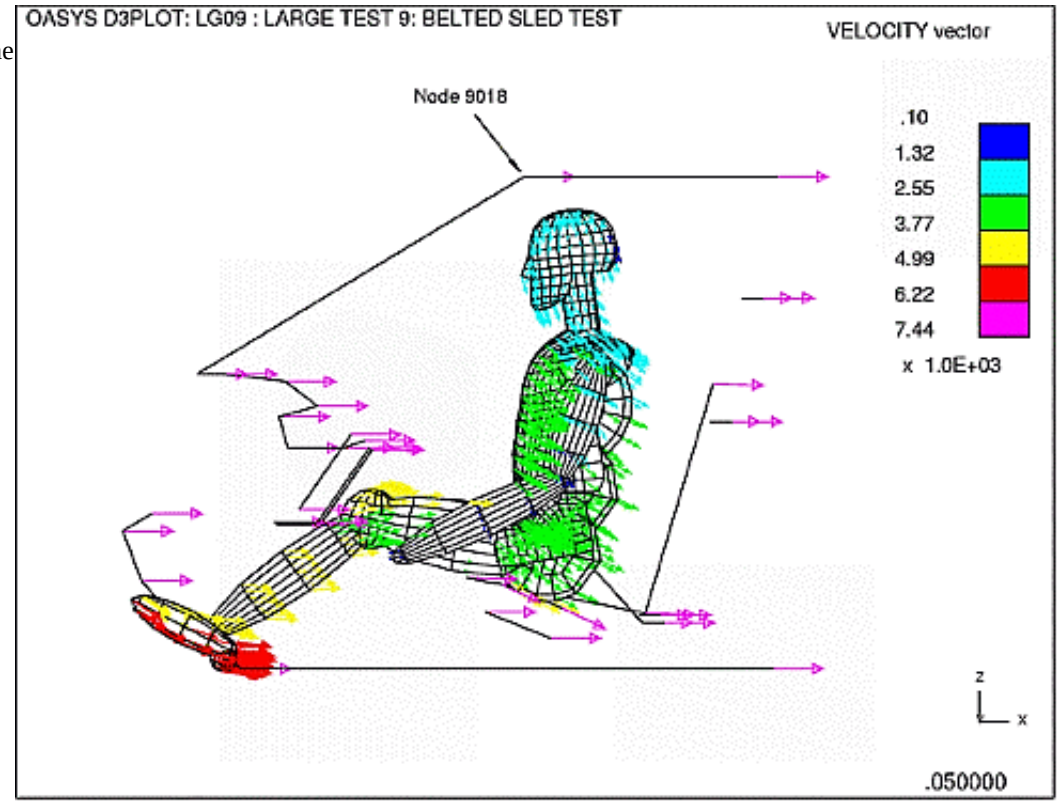
6.3.5.2 Using REFERENCE_NODE (single node case).

Here is an example showing how a single **REFERENCE_NODE** might be used.

In this case we have a dummy in a sled test, as above, where a crash is simulated by pulling the sled backwards. However what we are interested in is the velocity of the dummy relative to the sled, since in a real crash the sled (= car) would be more or less stationary, while the dummy would still be travelling forwards.

We can achieve this by picking a node on the (rigid) sled as our reference node, and displaying all velocities relative to that.

Here is the "raw" image, showing that the sled is moving rapidly backwards.

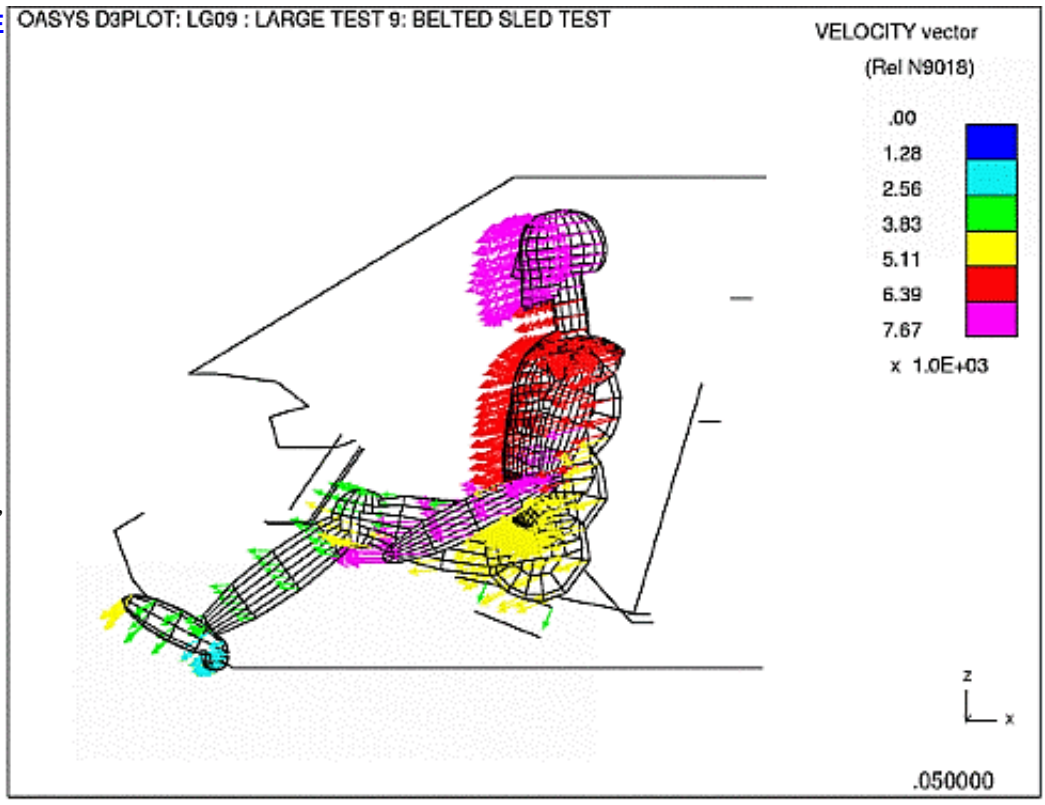


REFERENCE_NODE now switched on.

Here is the revised velocity plot now that the REFERENCE_NODE has been switched on.

The velocities of the sled at node 9018 have been subtracted from all velocities, making those on the dummy effectively relative to the sled.

(Should we wish to fix the sled in model space, and to draw the deformed shape of the dummy relative to that throughout an animation, we could also use FIXED_NODE. However the two operations are independent and do not have to be combined.)



6.3.5.3 Using REFERENCE_NODES (3 node case)

The following example shows how **REFERENCE_NODES** (3 nodes) works, and how it is related to **SHIFT_DEFORMED.**

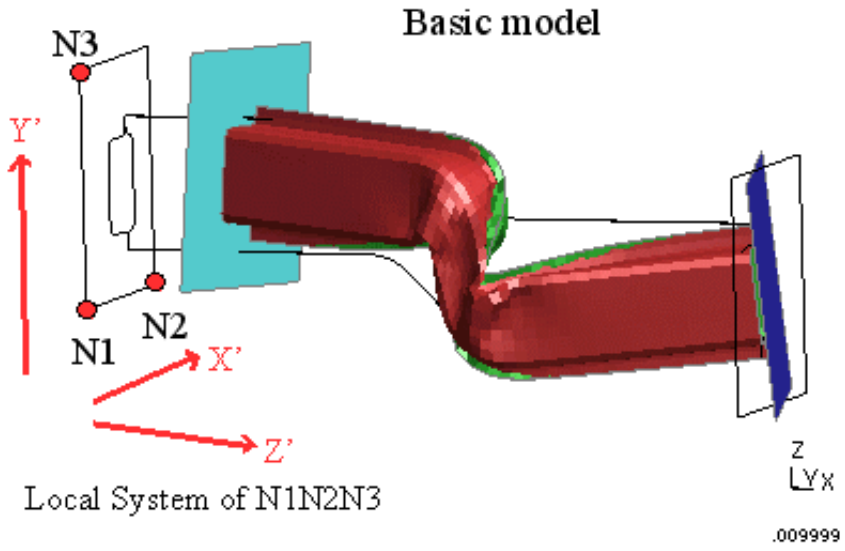
Here is the basic model.

It is a crush tube shown in its final state, with the undeformed geometry overlaid.

The loading platens at each end are pushed together, but they are free to rotate. The problem is to determine the maximum "end to end" deformation.

It is clear from this plot that the blue end moves and rotates, and this makes it difficult to determine the deformation relative to that end. The sequence below shows how to overcome this problem.

Three nodes (N1, N2, N3) have been chosen on the loading platen at the blue end, and they form a local coordinate system as shown.

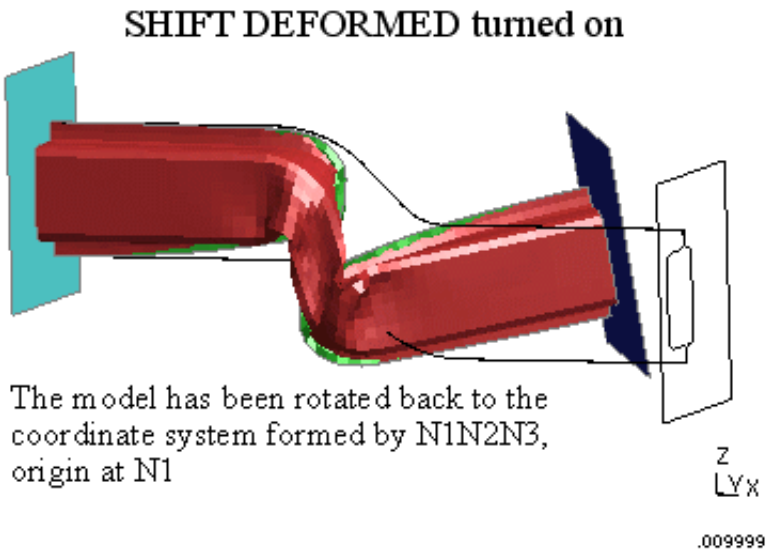


SHIFT_DEFORMED turned on.

This is the same model at the same state, but now SHIFT_DEFORMED has been switched on, and the model has been rotated back to the coordinate system formed by N1N2N3, translated back to origin at N1.

Note that the rotation and translation are back to the undeformed locations of nodes N1 to N3.

(*This step is not necessary in order to calculate data relative to reference nodes, but it makes the example much clearer.*)


SHIFT DEFORMED turned on
The model has been rotated back to the coordinate system formed by N1N2N3, origin at N1
Z
YX
.009999

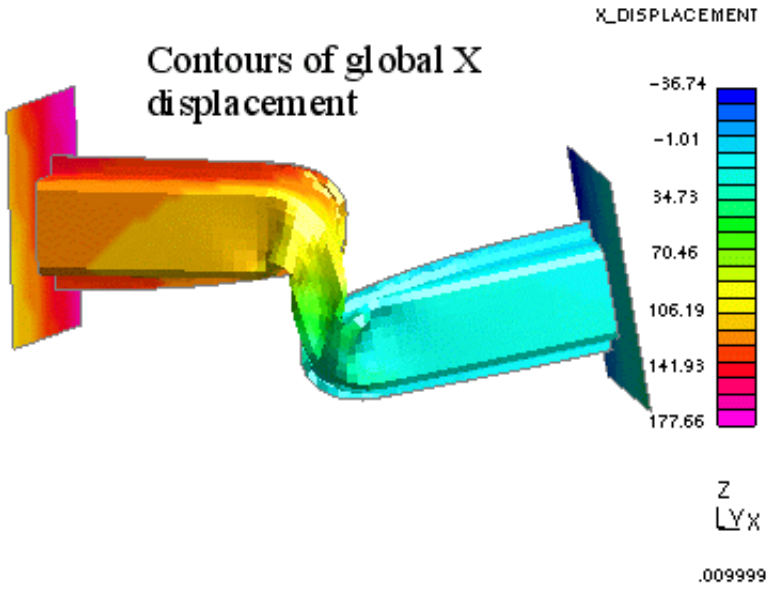
Contours of X displacement now shown.

This plot shows global X displacement, which is approximately along the length of the tube.

However because both ends of the tube have rotated it is difficult to estimate the movement of the two ends relative to one another. We can see that it approximately 177.66 + 36.74 = 214.4, but this may not be good enough.

In order to obtain a more accurate value it is necessary to express the displacements in terms of the coordinate system formed by N1N2N3.

Remember: SHIFT_DEFORMED ***only*** affects the deformations drawn, it has no effect on the values that are contoured or written out.

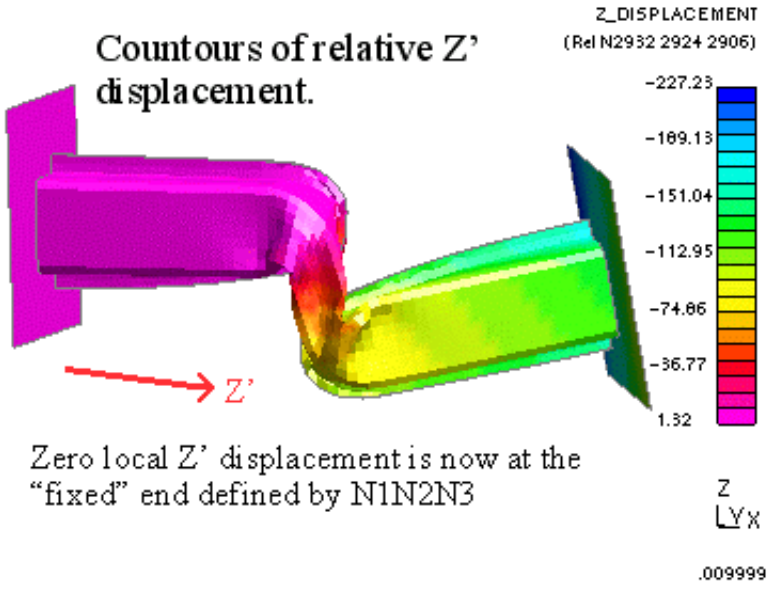


REFERENCE_NODES turned on, and contours of local Z' displacement shown

By switching on REFERENCE_NODES, and selecting output in the local system, we can now plot displacements in the local Z' direction relative to the left hand end.

It is now clear that the actual peak movement at end two is actually 227.23, somewhat higher than our estimate from the approximate global X plot above.

This technique is very useful when calculating "knock-back" and "intrusion" displacements at particular locations in a model.

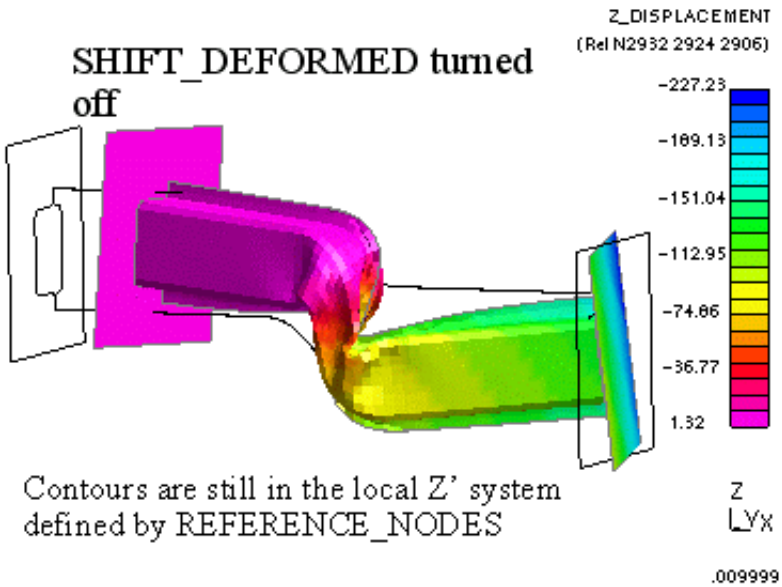


SHIFT_DEFORMED turned off, but REFERENCE_NODES left on.

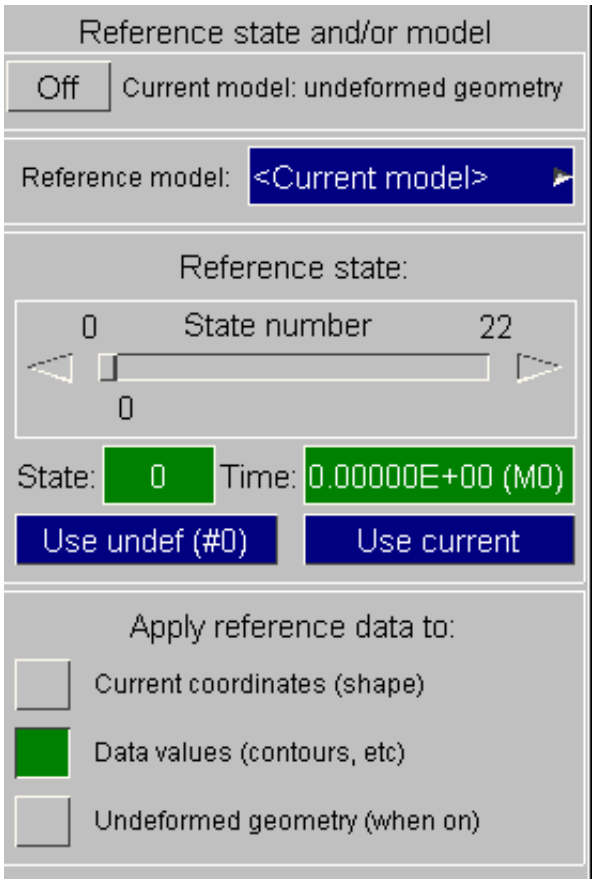
This plot demonstrates that while SHIFT_DEFORMED and REFERENCE_NODES are related, and share the same nodes, they can act independently.

SHIFT_DEFORMED has been turned off, so the deformed shape is now the "true" shape, but the contours are still expressed in the local Z' of the axis system defined by N1N2N3.

This is a harder plot to understand, because the axis system of the plotted results is not that easy to discern.



6.3.6 REFERENCE STATE/MODEL



Normally results at a given state are drawn and reported verbatim, subject to the various options above. However it is possible to subtract from the current data:

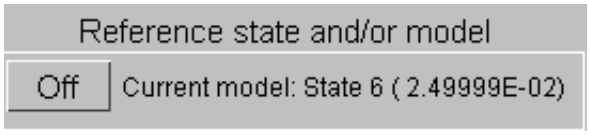
- The results at a different state in this model, showing the difference between two times.
- The results at the same, or a different, state in another model, showing the difference between models.

This operation is applied to all nodal and element data, and its application can be to any permutation of:

- The current graphical coordinates (ie the plotted shape)
- The current data values (ie contoured, written and time-history values)
- The current undeformed geometry plot.

By default the reference model is the current model, and the reference state is zero, and "reference plotting" for a window is turned off, meaning that no action is taken here.

6.3.6.1 Turning Reference Plotting on/off



Reference plotting is not active in any window until it is turned on in that window.

Settings for each window are stored separately, and this panel shows those for the first active window selected by the **W1 .. Wn** tabs. To define different settings in different windows select a single **Wn** tab at a time, and configure each window individually.

6.3.6.2 Choosing the Reference Model

Two possibilities exist:

Plot relative to the current model

This is the default case. You select a reference state in the current model, and results are plotted relative to this state. If you have multiple models in a window then each model will be plotted relative to itself.

Data from the reference state are subtracted from those in the current state, and the results displayed. It is perfectly possible to choose a reference state later than the current one, and hence to get "negative" results, the computation is simply:

$$\texttt{<data>}_{\texttt{displayed}} = \texttt{<data>}_{\texttt{current}} - \texttt{<data>}_{\texttt{reference}}$$

Plot relative to another model

This is the more complex case of plotting data from this model relative to a state from a different model. If you have multiple models in a window each model in that window will be plotted relative to the reference model.

The principle is exactly the same: the reference data is subtracted from the current, but mapping of reference model onto current is done as follows:

- **Mapping is "by external label".**

  The results from node label <i> in the reference are subtracted from those for node label <i> in the current model. And likewise for elements.
- **No checking for geometrical or topological proximity takes place.**

  No check is made that node <i> (or element <j>) in the two models are equivalent, or even remotely in the same place - either topologically or geometrically.
- **If no equivalent label is found, zero is reported.**

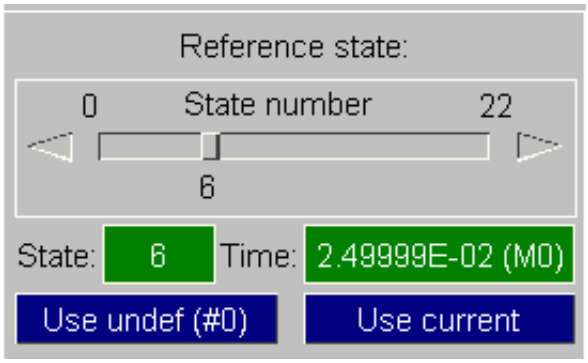
  If no matching node or element can be found in the reference model, then zero is reported as a result - regardless of the actual value in the current model.

This means that models which are topologically nearly identical compare well, but areas which have been remeshed may give very misleading comparisons.

6.3.6.3 Choosing the Reference State

Whether you are using the current or a reference model you must define which state in that model is to be used as the "reference" one.

You can use either a fixed state number, or a changing "current" one.



Using a fixed reference state

This is the default case, invariably used when plotting data relative to the current model.

You simply select a valid state from the model, and this is used as the reference state. It is perfectly legal to select a fixed state from a different model too.

Using the "current" state

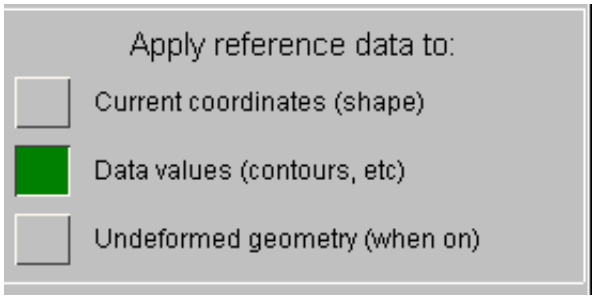
This meaningless if the reference model is the current model, since results will always be zero.

But if plotting relative to a different reference model it is a powerful tool, especially during animations:

For each state <i> the data from the equivalent state <i> in the reference model is subtracted.

This means that you can see how the differences between two models vary over time. Obviously it is important to make sure that the states in the two models have the same times, as no check is made for time equivalence.

6.3.6.4 Choosing what Reference data applies to.



You can control the extent to which "reference" logic applies to plotted geometry and contoured or written data.

Any permutation of the below can be selected.

Current Coordinates

Whether or not reference geometry is used for the shape that is plotted on the screen.

Note that "coordinates used for plotting" and the component "displacements" are kept separate, and can be controlled individually.

Data Values

These are the values contoured, vector plotted, written by the WRITE command and reported by XY_DATA.

Undeformed Geometry

This is the display of undeformed geometry on the plot only (when drawn). Normally this will always display state #0, but if you turn this on the geometry of the reference state will be used instead.

6.3.6.5 Examples of using Reference State/Model

The following examples demonstrate how the feature might be used.

Example 1: Reference state in the same model.

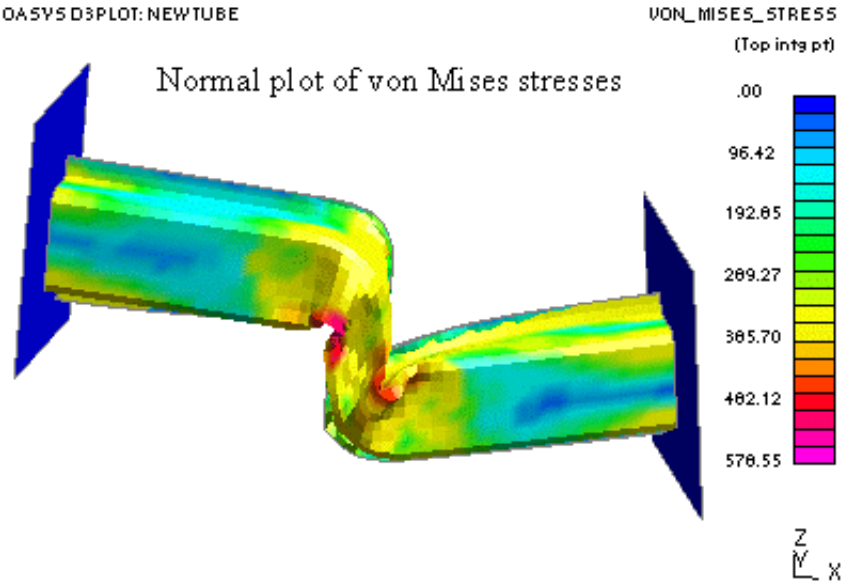
Here is the "raw" plot - Von Mises stress in a crush tube.

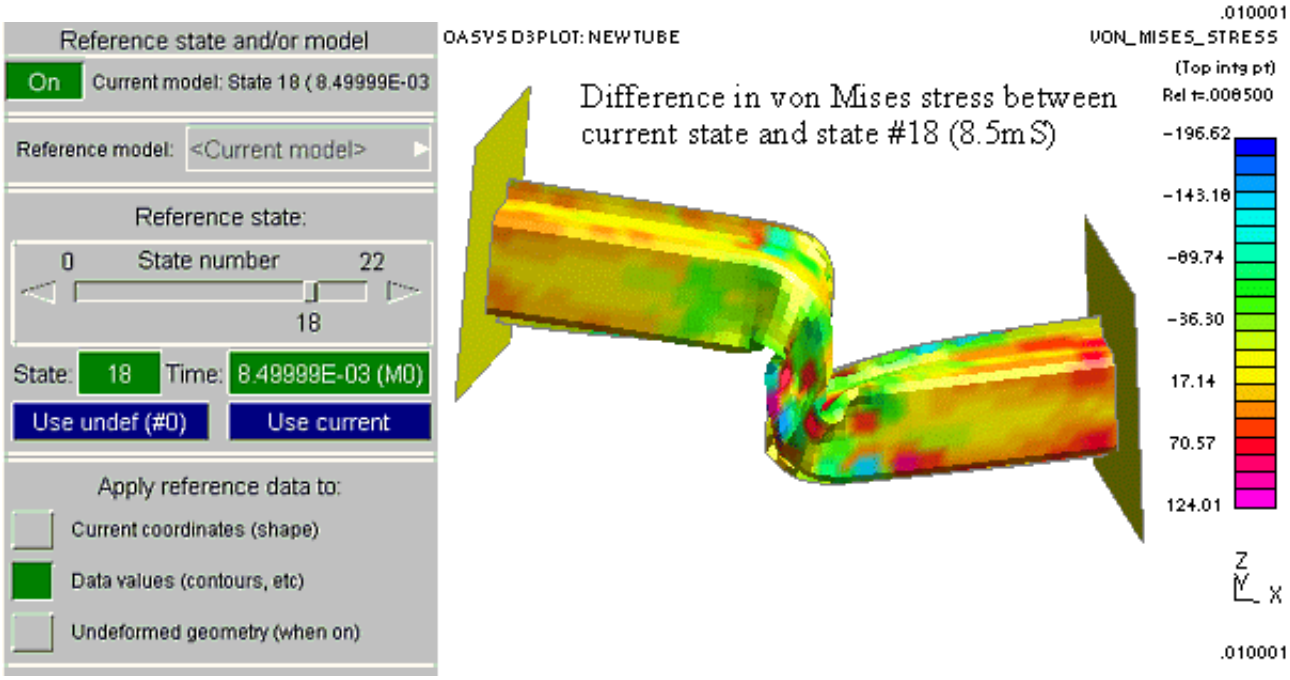
And below is the same plot with the reference state set to #18, which is at t = 0.0085s, in the same model. Remember that:

$<data>_{displayed} = <data>_{current} - <data>_{reference}$

Therefore, as in this example, negative von Mises stresses can be generated since, in some locations, the stress at an earlier state was in fact greater.

(Incidentally this illustrates that the reference calculation is performed upon the scalar output of the data component computation, not upon its individual tensor components.)



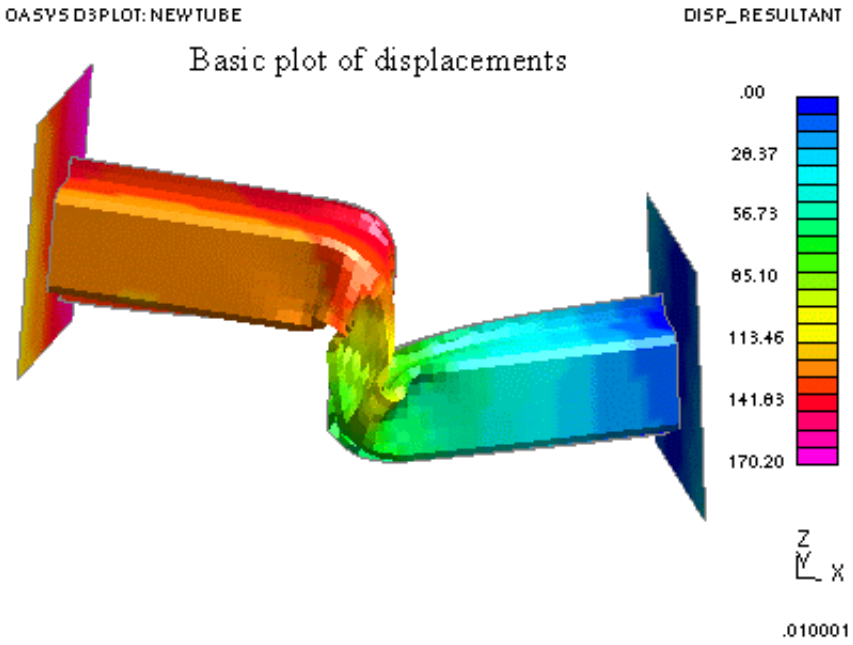


Example 2: Reference state in a different model

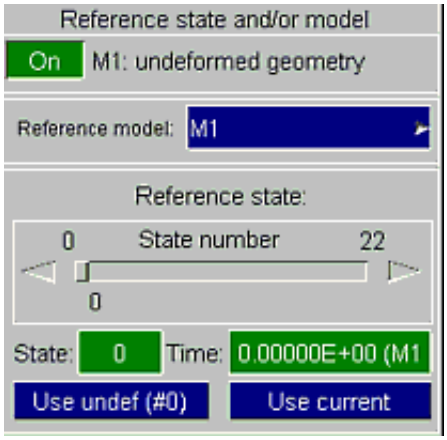
In this case the model above has been re-run, but with the section thickness of the crushable elements reduced by 25%. Here we are comparing the results between original and modified models to see what difference this makes.

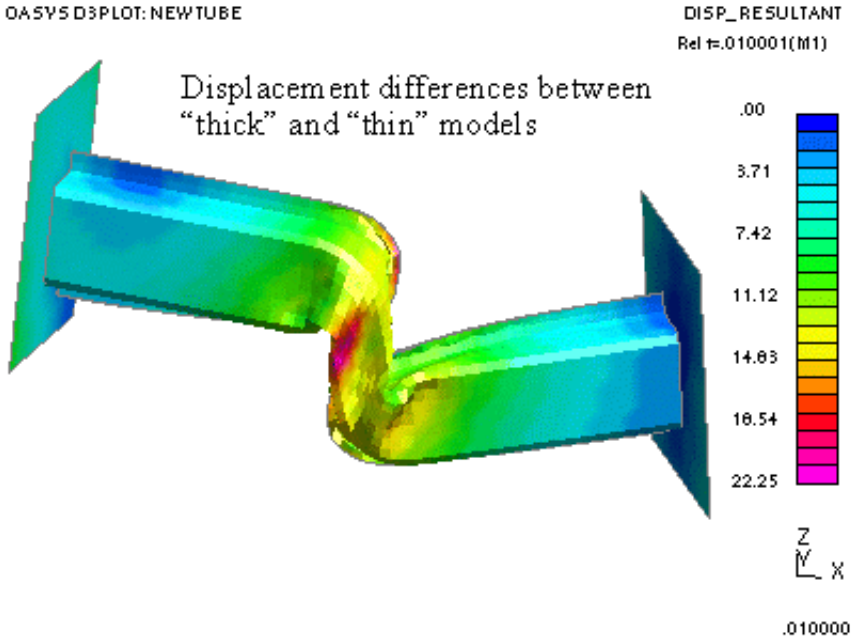
Obviously rerunning the same model with different section properties does not upset node or element labelling, so exact equivalence between the two analyses is preserved.

Here is the original image, showing contours of "true" displacement magnitude.



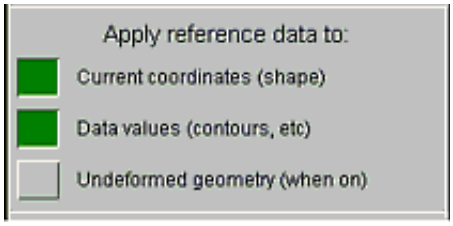
Here is the same image, with the displacements in the reference model subtracted from the "true" ones above. The "current" state has been used in the reference model (M1), and this is reported on the plot.

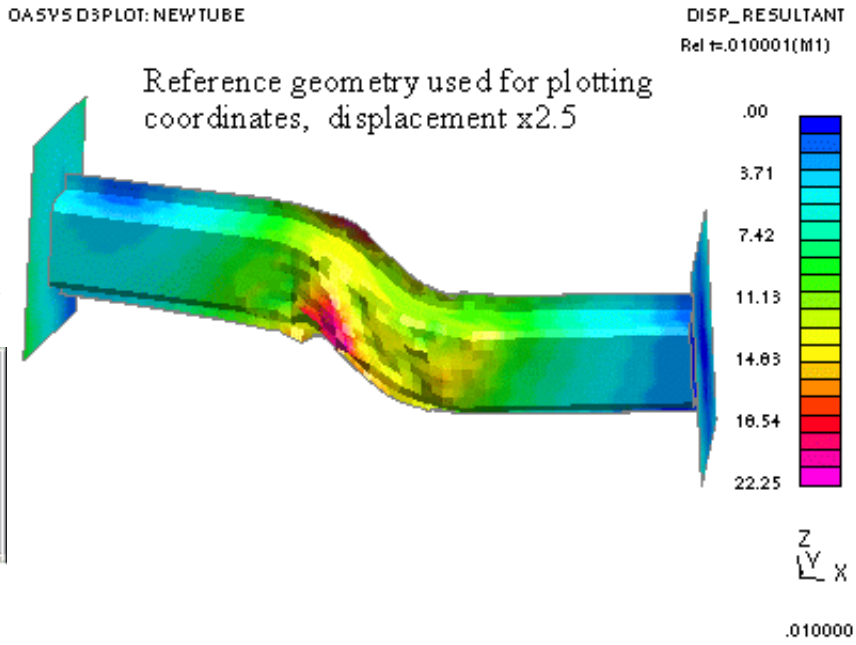




It is not that easy to visualise the differences in shape from the contour plot above, since they are obscured by the gross deformations of the structure.

So in this plot reference geometry has been used for the current (plotting) coordinates as well, showing the difference in displacement between the two models. Effectively this is the undeformed geometry + the difference in displacement between the two models.





Displacements have also been magnified by a factor of 2.5 using **MAGNIFY_DISPLACEMENTS** to exaggerate them, making them clearer.

REFERENCE STATE - Notes

1) If the reference state is set to state 0 then all values reported will be the absolute values.

2) If an analysis contains pre-stressed elements then state 1, not state 0, should be selected if values relative to the pre-stressed values are required. (State 1 is a genuine set of results at analysis time zero, state #0 is a synthesised set of zero values.)

3) The reference state option is not available for a model that includes adaptivity, see Section 4.1.11.

4) Using the reference state option will increase the amount of memory used by D3PLOT slightly, as two complete states have to be stored simultaneously.

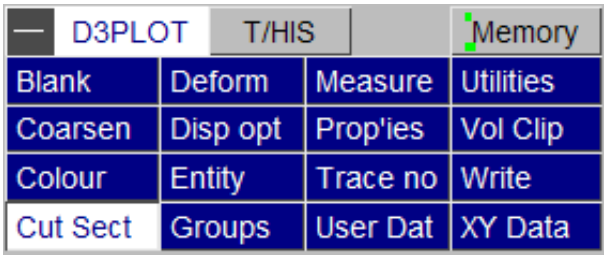
REFERENCE MODEL - Notes

1) Any model can be used as a reference model, but it should be reasonably similar to the original if sensible results are to be obtained.

2) Using a reference model will slow down plotting since the <current> vs <reference> lookup by label imposes an overhead. It can also slightly increase memory usage.

Next section

6.4 CUT_SECTIONS



The Cut Section menu is invoked from the Tools menu or from keyboard shortcut X.

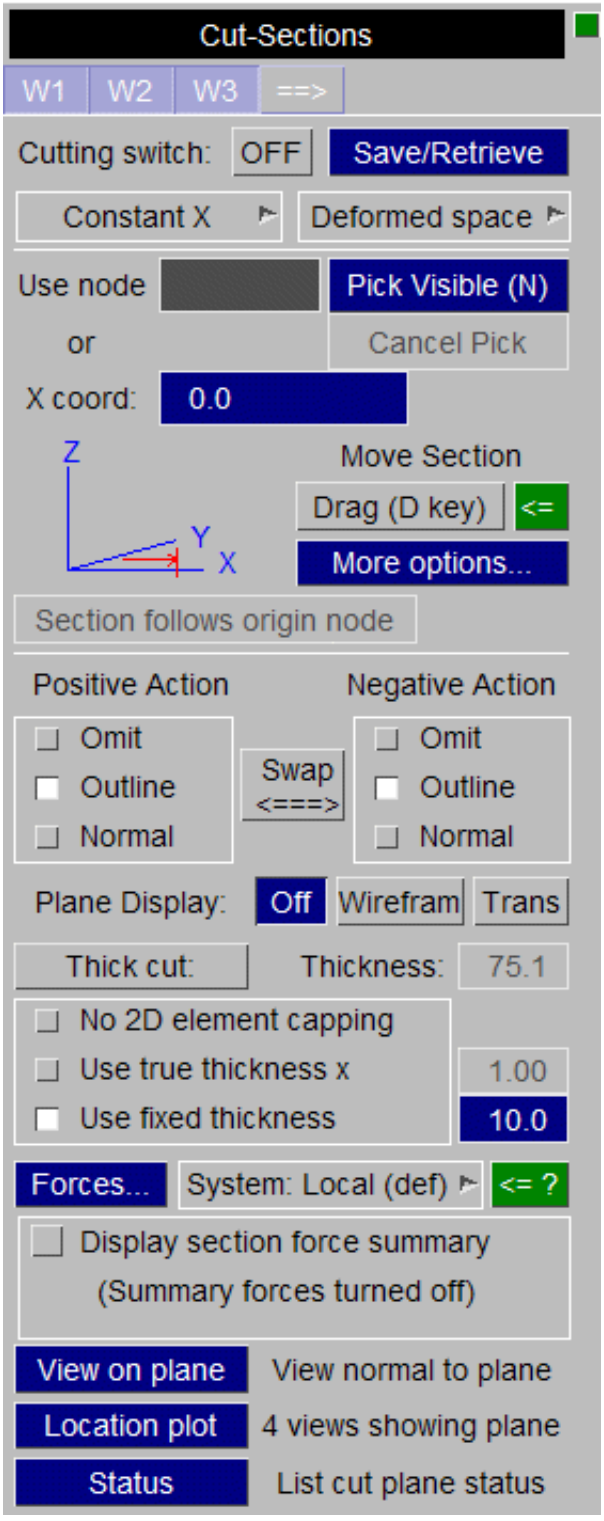
A cut-section, sometimes referred to as a "cutting plane", is a flat plane that cuts through the model. It may be located anywhere in space and oriented at any angle.

When the **Cutting switch** is turned on the intersection of the plane with the model is calculated and the interpolated cut plane is drawn.

This is possible in all D3PLOT display modes, (including animation), and for those that display data this will be displayed on the cut plane.

Various options, described below, define if and/or how the model either side of the plane is drawn.

The forces acting on the cut-plane, integrated from element stresses, may be calculated and output.



Cut Sections are a "per window" attribute

Cut-section definitions apply to all those windows which have their **`W1 .. Wn`** tabs set. They are stored as an origin coordinate and a local coordinate system, which cuts through all models in the relevant windows.

If you use the **PICK NODE ...** options to derive a coordinate from a node you will be forced to define which model to pick from, but thereafter the coordinate is model-independent.

If you use the option to track node motion across multiple models then special rules apply: see under **Section follows nodes** below.

6.4.1 Some important rules governing cut sections that must be clearly understood:

- Only one cutting plane can be current at any one time, although any number may be stored on disk and retrieved at will. The plane will only be active if turned on. By default (as with volume clipping) no plane is defined, and it is switched off.
- Only Solid, Shell, Beam and Thick shell elements are cut. Other element types, such as joints, springs, stonewalls, etc, are unaffected. You may want to remove these from the display when using cut planes since they will span the plane.
- Forces and moments on cut planes are also only calculated for elements of these four types which are unblanked, the others are ignored. This is because these are the only element types for which stress &/or force results are consistently available.
- Forces and moments are calculated from solid and thick shell stresses, shell force and moment resultants, and beam forces/moments. Therefore if any of these are rigid no forces will be computed for the relevant materials, even though the elements may be carrying load.
- **Forces** on planes are calculated reasonably accurately. **Moments** are only approximate and should only be treated as (usually under-) estimates.

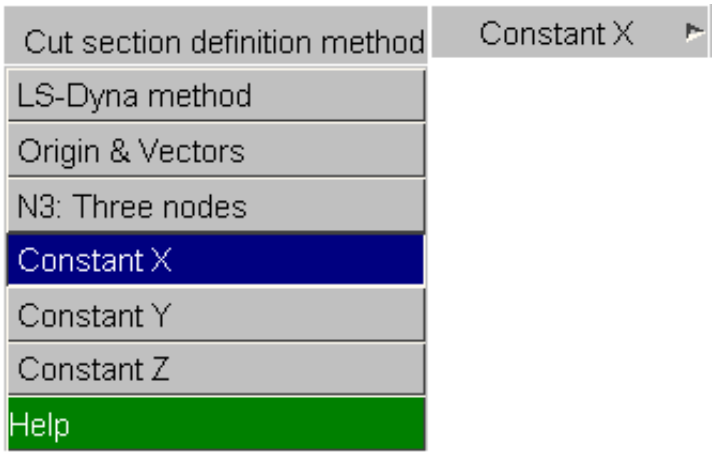
6.4.2 Creating a cutting plane

The first step in creating a cutting plane is to choose how you are going to define it.

A plane is defined by its origin and its local X', Y' and Z' vectors.

The top two options permit the section to be arbitrarily oriented in space, the lower three align it exactly with the model X, Y and Z axes respectively.

Regardless of how it is defined initially the internal definition of the plane is the same, and it may be translated and rotated at will later.



LS-DYNA Method

This option allows import of definitions in the format used by the LS-DYNA *DATABASE_CROSS_SECTION keyword:
- Normal vector tail coordinate
- Normal vector head coordinate
- Edge vector head coordinate

If you have written a .ZTF file from PRIMER than any database cross sections in your original input deck can be imported by using the **Import DATABASE_SECTION** option.

Note: LS-DYNA cross sections use ***lagrangian*** ("basic" in D3PLOT terminology) space.

When you define a cut section using this method you will be asked if you want to swap to "basic" space for compatibility with LS-DYNA. This is explained in more detail below in section 6.4.3. (See Appendix II for an oa_pref option that will allow you to set this as your default definition method.)

Origin and Vectors Method

This definition requires the user to enter (in model coordinate space) the:
- coordinates of the origin for the plane
- the local x-axis vector
- any vector lying in the local XY plane.

N3: Three nodes method

This method requires you to pick three nodes which form the local axis system as follows:
- Node 1 is the orign
- Node 2 gives the local X axis from the vector |N1N2|.
- Node 3 gives the local X axis from the vector |N1N2|.

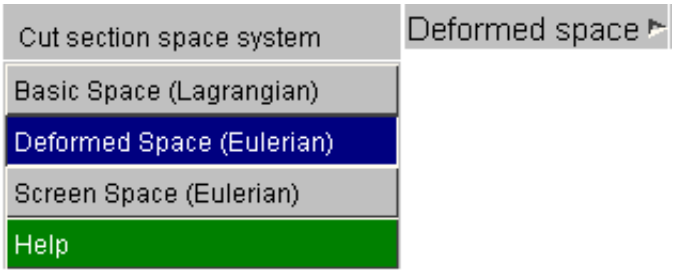
Normally the coordinates of the nodes at the current state form the basis of the plane definition, but if you choose "Section follows nodes" you can update the plane at every state as the nodes move.

Constant X,Y,Z Method

The three "constant" values allow you to define the coordinate along the model X, Y or Z axes respectively at which a plane of that constant axis value will be defined. Locally:
- The origin will be [0,0] on the other two axes.
- Local Z is in the +ve direction down the axis chosen.
- The other two axes are chosen for you, aligned with the two unchosen model axes.

If you use a node here then normally the coordinates of that node at the current state are used, but if you choose "Section follows nodes" you can update the plane at every state as the nodes move.

6.4.3 Defining a space system for the plane.



Once you have defined the plane, by one of the definition methods above, you need to define which space system it operates in.

This figure shows the Cut space system selection panel, showing the three possible systems. These are described below.

Section follows node(s) allows a cut section defined using 3 nodes, or a single node in the constant X/Y/Z cases, to be updated using the current coordinates of the node(s) at each state.

BASIC space system

In this system the cut plane is calculated using the model's **undeformed** geometry, regardless of the current state in core.

This means that the parametric coordinates of the cut positions on elements are calculated using the undeformed geometry, then applied to the current (deformed) in-core state. Therefore the cut plane will almost certainly not remain flat as the model deforms.

This is a "lagrangian" cut: the cutting plane deforms as the element mesh deforms.

DEFORMED space system

In this system the cut plane is calculated using the model's current **deformed** geometry. Therefore the cut position on elements, and indeed the elements which are cut, can change as the model deforms through the static plane position.

The plane will always remain flat, and will remain fixed in space relative to the model coordinate system.

This is an "eulerian" cut: the cutting plane remains fixed while the element mesh can deform through it.

SCREEN space system

In this system the cut plane is calculated using the current screen coordinates, after the transformation and projection to screen space.

This has the effect of tying the cutting plane to the screen space system, effectively to your display, therefore both deformations and viewing transformations (eg dynamic viewing) can move the model through the plane.

This is also an "eulerian" transformation since the model deforms through a static cutting plane.

Note: Force and moment computation varies with section space.

For compatibility with LS-DYNA the forces and moments computed in a BASIC space system are:
- Always expressed in the global cartesian system
- Centred on the average coordinate of the cut section at each state.

Whereas those computed in a DEFORMED or SCREEN space system are:
- Always expressed in the section local coordinate system.
- Centred on the plane origin as defined by the user.

This is described in more detail in section 6.4.4 below.

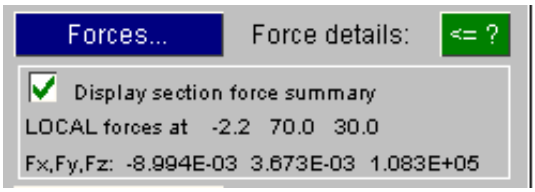
Section follows Node(s)

If a cut section has been defined using either the 3 node method or a single node and a global axis system then this option will force the cut section to follow the node(s) as they move during the analysis.

Where there is more than one model in the window(s) affected then the following rules apply:
- For each model the labels of the nodes are looked up.
- If all the necessary nodes (3 in "3 nodes" case, 1 in "constant X/Y/Z" case) are found then the normal logic will apply based on the current coordinates of the nodes in each model, and the section will be updated at every state.
- If a node is not found then the "follow" logic is turned off for that model, and the plane will remain static in its initial position for that model.

Note that using this logic over multiple models may mean that the planes in each model may not be the same, as the defining nodes may move differently. Exercise care using this option!

6.4.4 **FORCES** Computing forces and moments on the cutting plane



This command calculates the forces and moments acting on the current current plane. The **<-?** button gives specific on-line help on this subject.

A summary of the section centre coordinate and the current forces on it can be shown in the master panel if the check box there is ticked. These figures will update automatically as the section is dragged.

Please read the following section on force and moment extraction before using this facility. There are some less than obvious pitfalls that you need to consider. In particular:
- Only ***unblanked*** elements are included in cut section force calculation.
- Rigid elements may be transmitting force, but they will always report zero output to the database. Therefore their contribution to cut forces and moments will always be zero.
- Forces and moments are only computed from Solids, Beams, Shells and Thick shells. Other element types either do not report forces (eg springs, seatbelts), or are not sensible in this context (eg SPH elements).
- There are inconsistencies in the way LS-DYNA writes beam force and moment output prior to LS971, requiring user intervention if the correct answers are to be calculated.
- Local bending moments in thick shells are not included, and may also be omitted for thin shells if force & moment resultant data components are not present in the database.
- LS-DYNA processes cut-sections in "basic" space, generating forces and moments in the global system. If you switch to "basic" space in D3PLOT you will get a similar calculation, but "deformed" space results in D3PLOT are expressed in the plane's local system.

How cut forces are calculated.

Only forces in the following ***unblanked*** element types are computed: solids, (thin) shells, thick shells and beams. Other element types (eg springs) are ignored since LS-DYNA does not report forces in them in a way that can be read by D3PLOT. The force and moment values are integrated from the element stress & force results as follows:

Solids: The cut face through the solid is interpolated, and its area calculated. The element stress tensor is rotated to the cut plane system and the forces are calculated from:

$$F_X = \tau_{XZ} * Area$$
$$F_Y = \tau_{YZ} * Area$$
$$F_Z = \sigma_Z * Area$$

Where: Fx is in-plane X force
Fy is in-plane Y force
Fz is normal Z force

No local element moments are calculated within solids: they are constant stress elements.

Thin shells: The shell is expanded into its true 3D shape by adding its (thickness/2) either side of its neutral axis. The cut area is then calculated in the same way as for a solid element.

The forces are calculated using the shell force resultants [Fx,Fy,Fxy,Qxx,Qyy] which, when divided by thickness, yield a stress tensor in the shell local coordinate system. This tensor is rotated to the cut plane system and forces are calculated as for solids above.

Local element moments are also obtained by rotating the moment resultants [Mx,My,Mxy] to the cut plane axes. But these are only approximate.

If shell force and moment resultants are not present in the database then the element stress tensor is used instead. However ***only the mid-surface results are used***, treating the element as a quasi-solid, which means that ***element local bending moments are ignored***. (*If the location of the element integration points and the degree of plasticity are both unknown it is impossible to calculate an accurate local bending moment.*)

Thick shells: The cut face through the element is calculated in exactly the same way as for a solid, yielding an area. The ***middle*** surface stress tensor is then applied over this area to give element forces as for solids above.

It is not possible to calculate thick shell moments properly since no moment resultants are available, and the distribution of bending stress through the element is unknown. So no local moments are calculated for these elements.

Beams: The three beam forces [Fx,Fy,Fz] and moments [Mxx,Myy,Mzz] are treated as constant vectors, are rotated to the cutting plane system and used directly.

Inconsistent beam sign conventions in LS-DYNA releases up to and including 970

Due to a bug in LS-DYNA versions up to and including LS970 exhibit the following inconsistent sign convention for beam output:

- "Resultant" (typically Belytschko-Schwer) elements use one sign convention
- "Integrated" (typically Hughes-Liu) elements use the opposite sign convention for 4 of the 6 output components.

The following table shows the sign conventions from releases 970 and earlier:

Component	Matching?
Fx	Same
Fy	Opposite
Fz	Opposite
Mxx	Opposite
Myy	Opposite
Mzz	Same

Sadly there is no "right" convention for beam output, as different users have different conventions. The confusion arises because of the different ways in which the beam types work: integrated beams have integration points at their centre, whereas resultant beams have (potential) hinges at their ends. The former reports force in the beam, and the latter reactions at the supports.

D3PLOT attempts to draw bending moment diagrams on the tensile side, but depending on which beam type you have used this may or may not be the case.

Beam sign conventions are consistent from LS-DYNA release 971 onwards

At some stage during the development of LS971 this problem was fixed, and results now use the "integrated" convention for all beam types. This is consistent with the reporting method for other element types in LS-DYNA, where results are the forces and moments within the element.

How D3PLOT handles the beam sign convention problem.

The sign convention is crucial when computing cut forces, since the force and moment vectors are transformed into the plane of the cut, and a reversal of their sign obviously affects the answers.

Unfortunately D3PLOT can’t tell whether an output database is from LS-DYNA 971 or later, since although the database contains a "version" field LS971 writes "970" in there! Therefore it doesn’t "know" which sort of beam it is dealing with and it will ask you what beam types you have used when you first calculate cut forces through a structure. Thereafter it will apply correction factors as required . If you have mixed the two beam types in your model you will have to be extremely careful when interpreting results from a pre-970 analysis.

WARNING: **Rigid elements report zero stresses, although they may still be transmitting loads.** The cut forces in these elements will be calculated as zero.

Sign convention of cut forces.

The sign convention adopted is that of the forces acting on the plane from its positive side, expressed in:
- **Deformed space**: the plane's local coordinate system.
- **Basic space**: the global model system

Moments are expressed about an axis, using the right hand screw rule. Thus Mxx is moment about the plane X-X axis in the relevant system. This is illustrated graphically in the figures below.

Coordinate system and centroid of cut forces.

The coordinate system and centroid depend upon the plane's space system:

<table>
<tr><td colspan="3">In DEFORMED space the following is used, all axes being in the plane local axis system.</td></tr>
<tr><td>Fx is in plane X force (effectively shear force)</td><td>Mxx is moment about plane local XX axis</td><td rowspan="3">The plane centroid is at the plane's origin, and its local X, Y and Z axes are as defined by the user. These axes may be visualised by turning plane display on. Unless "cut follows nodes" is turned on the centroid and axes remain fixed as the model deforms.

In particular note that moments are calculated about plane local axes through the plane origin.</td></tr>
<tr><td>Fy in is plane Y force (also shear force)</td><td>Myy is moment about local YY</td></tr>
<tr><td>Fz is Z force normal to the plane.</td><td>Mzz is moment about local ZZ</td></tr>
</table>

<table>
<tr><td colspan="3">In BASIC space the following are used, all axes being in the global model system.</td></tr>
<tr><td>Fx is force in the global X axis</td><td>Mxx is moment about the global XX axis</td><td rowspan="3">The plane centroid at any given state is the average coordinate of the cut elements, this means that it moves as the model deforms.

In particular note that moments are calculated about plane global axes through the current plane centroid as calculated from the average of all cut elements.</td></tr>
<tr><td>Fy is force in the global Y axis</td><td>Myy is moment about the global YY axis</td></tr>
<tr><td>Fz is force in the global Z axis</td><td>Mzz is moment about the global ZZ</td></tr>
</table>

Compatibility with *DATABASE_CROSS_SECTION output from LS-DYNA

LS-DYNA uses the lagrangian approach for cross-sections, and computes their forces using the equivalent of the BASIC method above. Results from LS-DYNA should match those from D3PLOT when BASIC space is used..

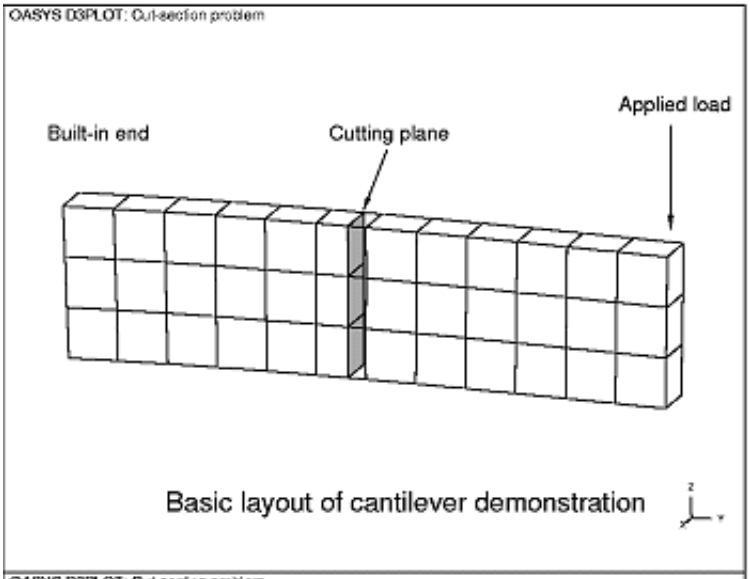
Consider the following example of a cantilever cut along its length:
In this example a cantilever made of solids is loaded downwards at its free end.

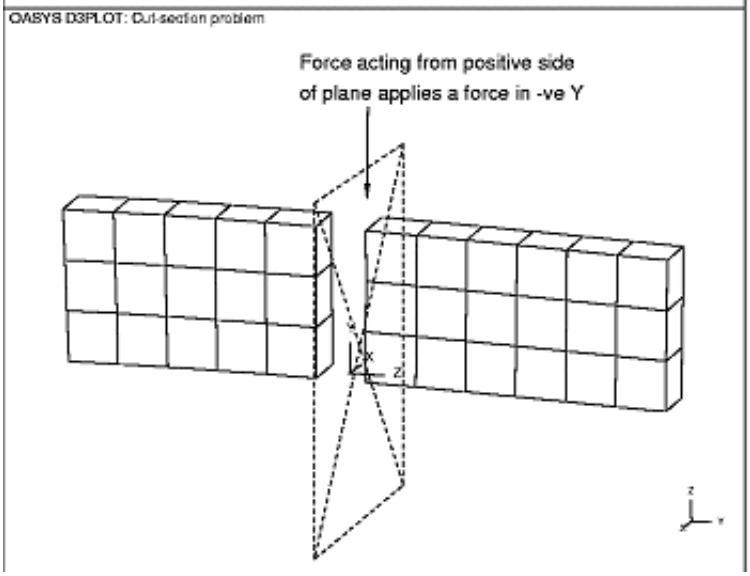
There is a cutting plane defined in DEFORMED space roughly half way along its length, with the positive side (+ve Z axis) being the free end.

The force acting on the cut plane from its +ve side acts downwards.

In the plane local coordinate system this is in the negative Y sense.

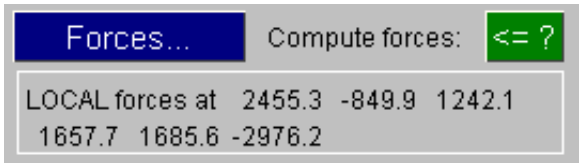
The moment acting on the plane is in the positive sense about the local XX axis.





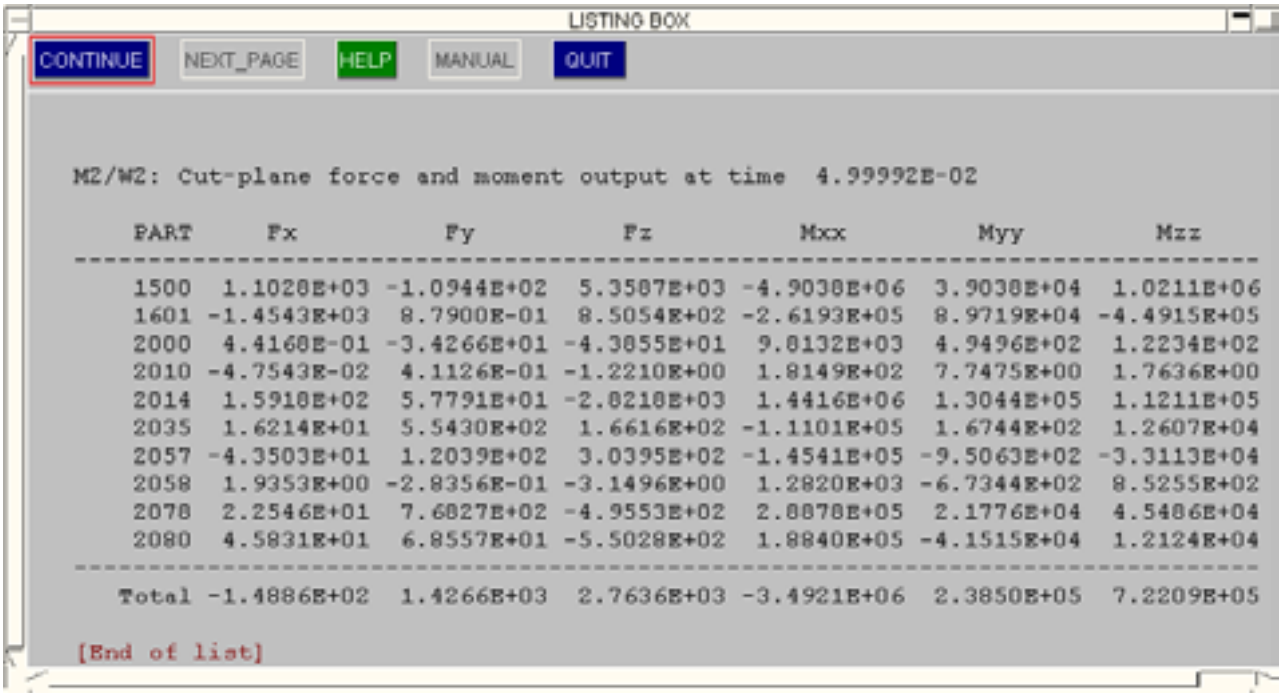
Methods of obtaining *written* force output

Method 1: Instantaneous force output on Cut-section menu



Method 2: FORCES... Command in the Cut-sections menu

Output here lists the 3 forces and 3 moments broken down for each material that actually contributes force, together with a summary for the whole model. Remember that elements which are blanked are not included, and materials which do not contribute any force are also not reported in order to keep the list short. An example is shown below:



M2/W2: Cut-plane force and moment output at time 4.99992E-02

PART	Fx	Fy	Fz	Mxx	Myy	Mzz
1500	1.1020E+03	-1.0944E+02	5.3587E+03	-4.9030E+06	3.9030E+04	1.0211E+06
1601	-1.4543E+03	8.7900E-01	8.5054E+02	-2.6193E+05	8.9719E+04	-4.4915E+05
2000	4.4160E-01	-3.4266E+01	-4.3855E+01	9.0132E+03	4.9496E+02	1.2234E+02
2010	-4.7543E-02	4.1126E-01	-1.2210E+00	1.8149E+02	7.7475E+00	1.7636E+00
2014	1.5918E+02	5.7791E+01	-2.8218E+03	1.4416E+06	1.3044E+05	1.1211E+05
2035	1.6214E+01	5.5430E+02	1.6616E+02	-1.1101E+05	1.6744E+02	1.2607E+04
2057	-4.3503E+01	1.2039E+02	3.0395E+02	-1.4541E+05	-9.5063E+02	-3.3113E+04
2058	1.9353E+00	-2.8356E-01	-3.1496E+00	1.2820E+03	-6.7344E+02	8.5255E+02
2078	2.2546E+01	7.6827E+02	-4.9553E+02	2.8878E+05	2.1776E+04	4.5486E+04
2080	4.5831E+01	6.8557E+01	-5.5028E+02	1.8840E+05	-4.1515E+04	1.2124E+04
Total	-1.4886E+02	1.4266E+03	2.7636E+03	-3.4921E+06	2.3850E+05	7.2209E+05

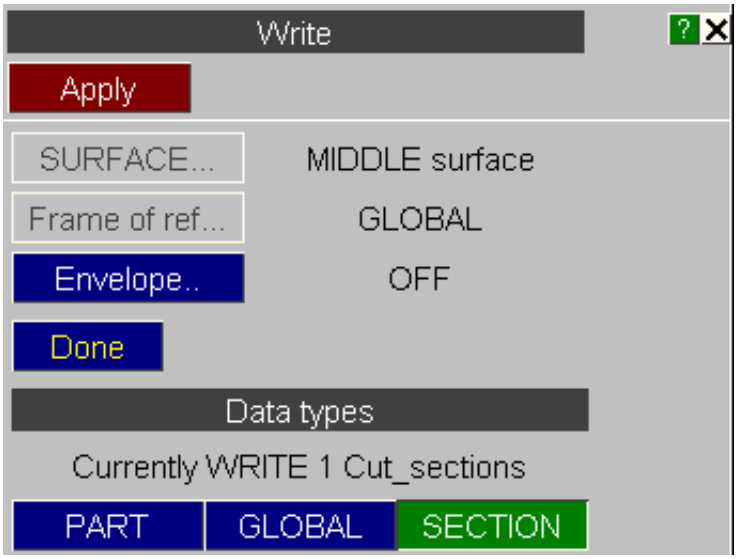
[End of list]

Method 3: WRITE Output (as an entity type)
When a cut-section is current the **SECTION** entity will be available as an "entity" type in the **WRITE** menu:

To use this select **SECTION** and then a data component.

You can select a scalar component, (such as **FX_X_CUT_FORCE** as shown here), or a summary of all forces and moments.

The advantage of using **WRITE** is that the results can be directed to file for subsequent use. An example of such output is given below:




```
++++++++ Data at time .30046E-02 ++++++++
CUT_SECTIONs:listing of CS_CUT_SUMMARY
Cut section: Fx                    Fy                  Fz                  Mxx
-------------------------------------------------------------------------------------------------------------------
Section      -9.667E+03            1.290E-04           -2.004E+00          -5.770E+01
```

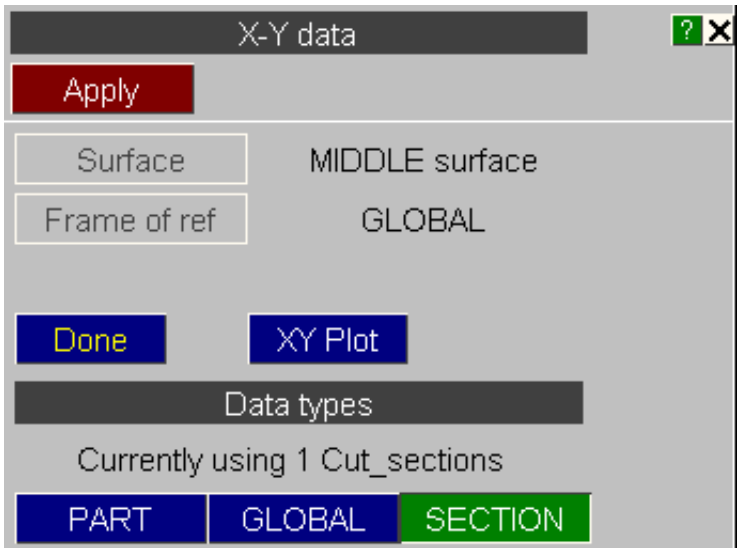
Obtaining XY graphs of cut-section forces wrt. time.

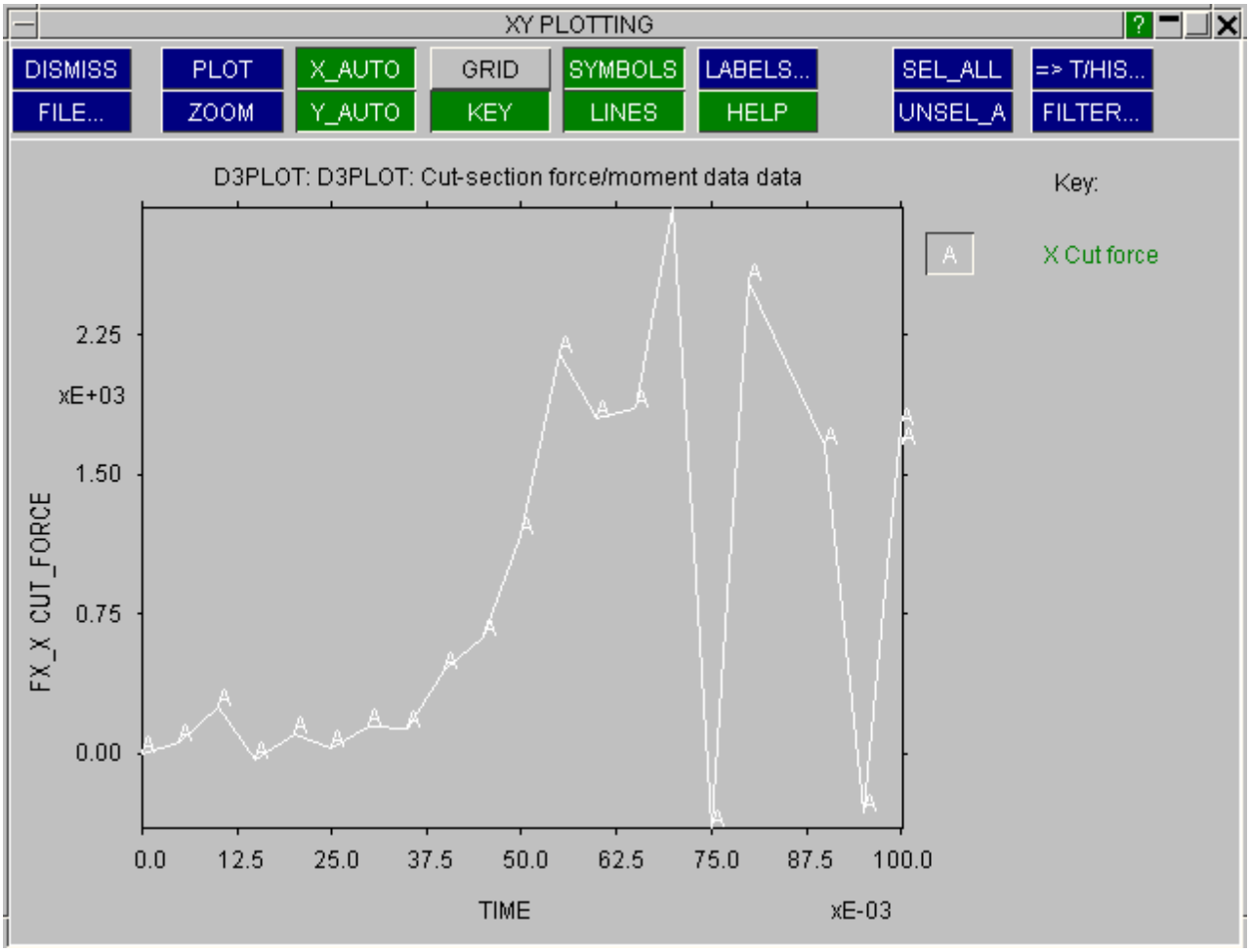
It is possible to use cut-section forces as a valid entity type for output in the **XY_PLOT** command, in a fashion similar to **WRITE**. When there is a current cut-section the entity type **SECTION** will be available in the **XY_PLOT** panel.

Choose the **SECTION** "entity" type, followed by a scalar data component, and the results of that component versus time will be generated.

Results can be written to disk as "curve" files, using the name **`sect001.cur`** etc, and also drawn as graphs in D3PLOT.

The figure below shows typical output of X cut force vs. time.





Note that you can also write plotted results to a curve file using the **FILE...** command on the plotting panel. (See Section 6.8 for more details.)

Cut area and centroid

It is also possible to extract data components **CUT_AREA** and **CX/Y/Z_CENTROID** of the cut section for display in **WRITE** and **XY_DATA**.

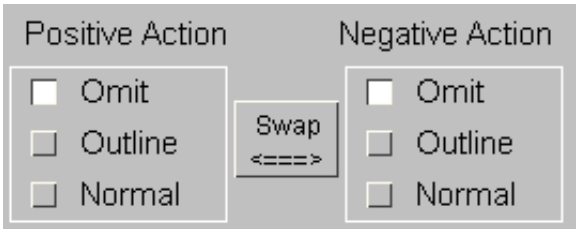
CUT_AREA Is the sum of all **Solid**, **Shell** and **Thick shell** cut face areas. No attempt is made to compute the area of cuts through beams.

CX/Y/Z_CENTROID Is the average coordinates of the cut face, computed from the 1st moment of area of the cut elements areas. Therefore it suffers from the limitation above in that it does not include beam elements.

Note that this is the case for both **Basic** and **Deformed** space sections, and in the latter case the reported centroid may be very different to the section origin (although it will be on the section plane).

Next section

6.4.5 **Positive & Negative Action** Controlling display of structure either side of the cutting plane.



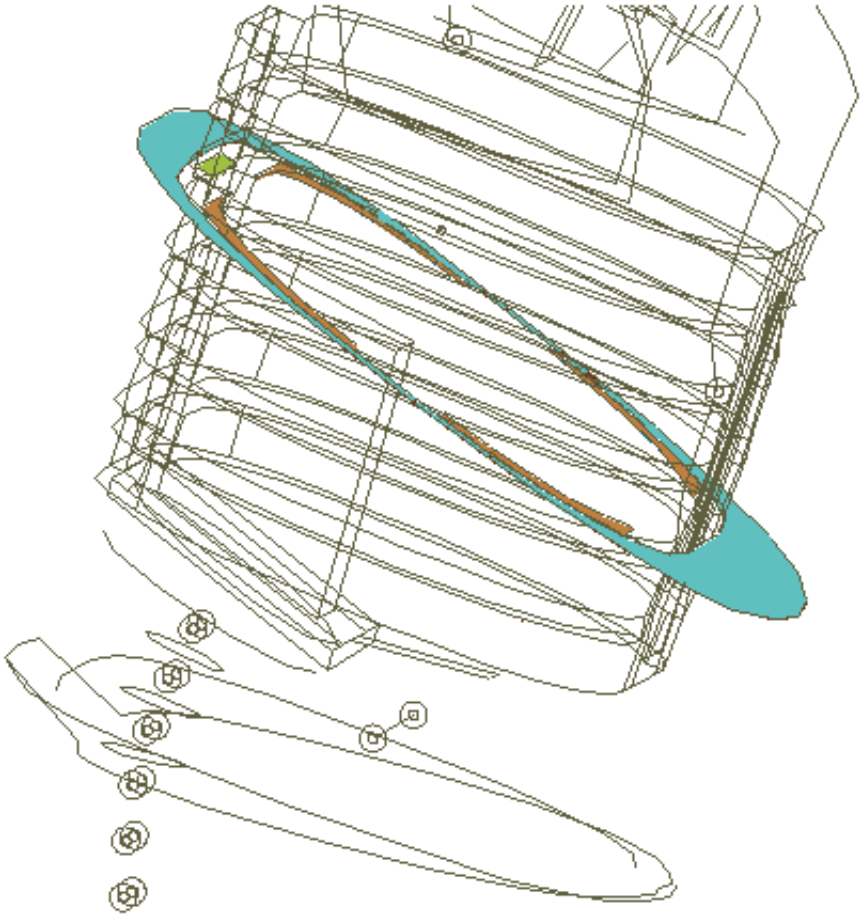
By default when cutting planes are switched on only the cut elements are drawn normally, with the remainder in wire-frame.. But it is possible to draw the mesh on both positive and negative sides of the plane at three levels of complexity. Each side can be controlled separately.
The options are:

OMIT The mesh on this side is not drawn at all.

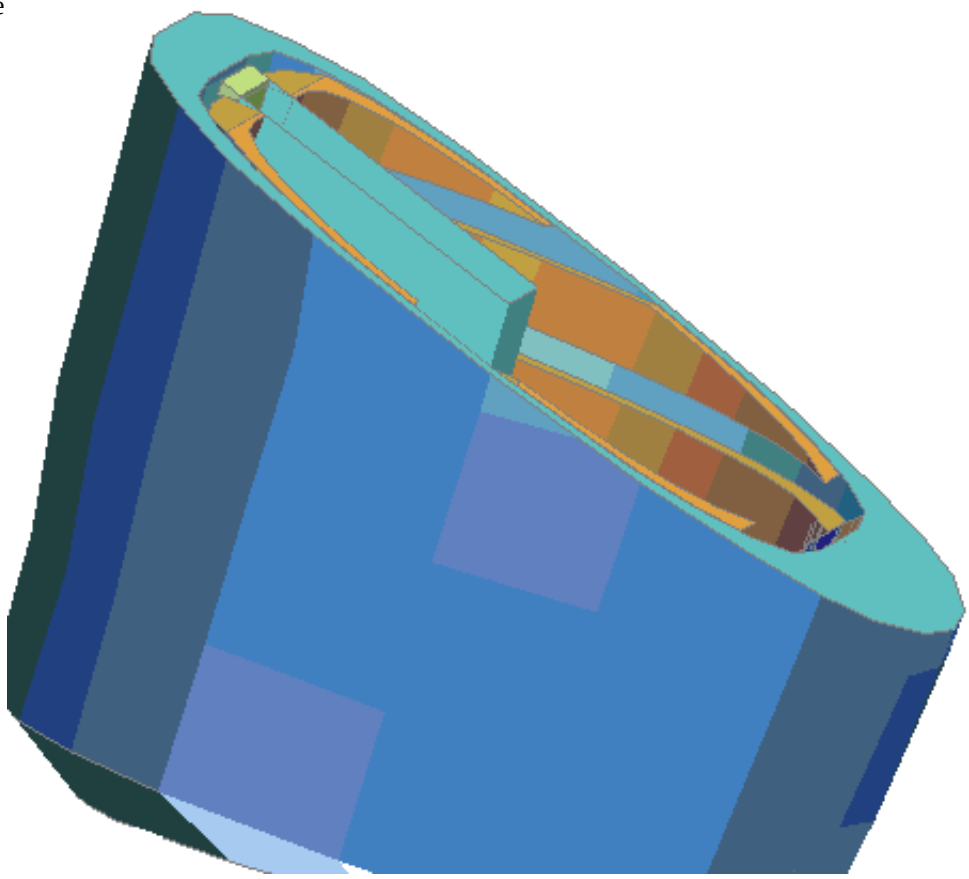
OUTLINE The mesh is drawn in "line" mode. This means no hidden-surface removal, and the cut plane will be visible through the mesh.

NORMAL The mesh on that side is drawn normally, with contoured data if applicable. Contours will be continuous over cut and uncut faces.

This image shows a cut section with the default settings(with both sides in outline mode)

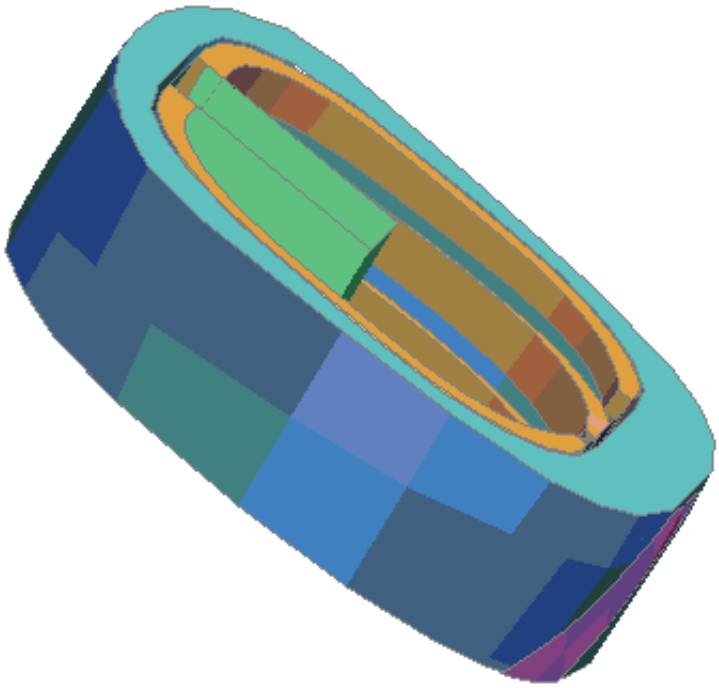


This image shows a cut section set to Omit on the positive side of the cut and Normal on the negative.



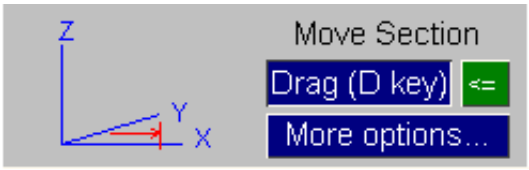
6.4.6 **THICK CUT** Creating cut sections with a finite thickness

Normal cut sections represent an infinitely thin slice through a model. The **THICKNESS** option can be used to generate a finite thickness cut through a model.



A 75mm thich cut

6.4.7 Dragging a plane interactively using the mouse.



The "**Drag**" button (or the "D" keyboard shortcut) switches D3PLOT into cut-section plane dragging mode.
In this mode the cursor symbol changes to "Sect Drag", and the mouse buttons act as follows:

- Left mouse ***translates*** the plane in the normal (local **Z**) direction.
- Middle mouse ***rotates*** the plane about the plane local **X** axis
- Right mouse ***rotates*** the plane about the plane local **Y** axis.

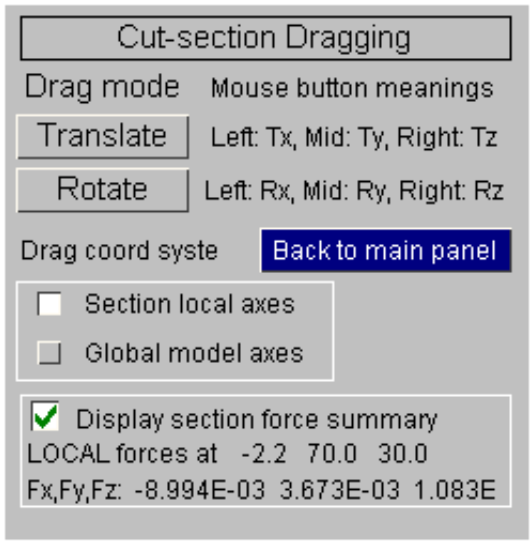
These operations are chosen because they are the most commonly used "drag" functions.

The **More options** button gives a more comprehensive plane dragging sub-menu.

You choose either Translate or Rotate, and in each case
- Left mouse translate along / rotates about the **X** axis
- Middle mouse ... ditto ... about the **Y** axis
- Right mouse ... ditto ... about the **Z** axis.

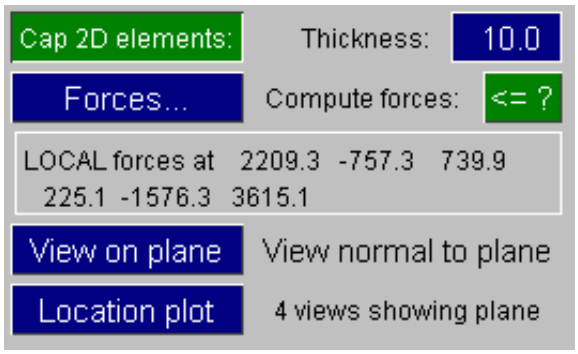
You can choose whether these translations / rotations use plane local or model global axis systems.

Cut section summary forces are shown as in the master panel and, if selected, will be updated as the plane moves.



6.4.8 Other cut-section commands

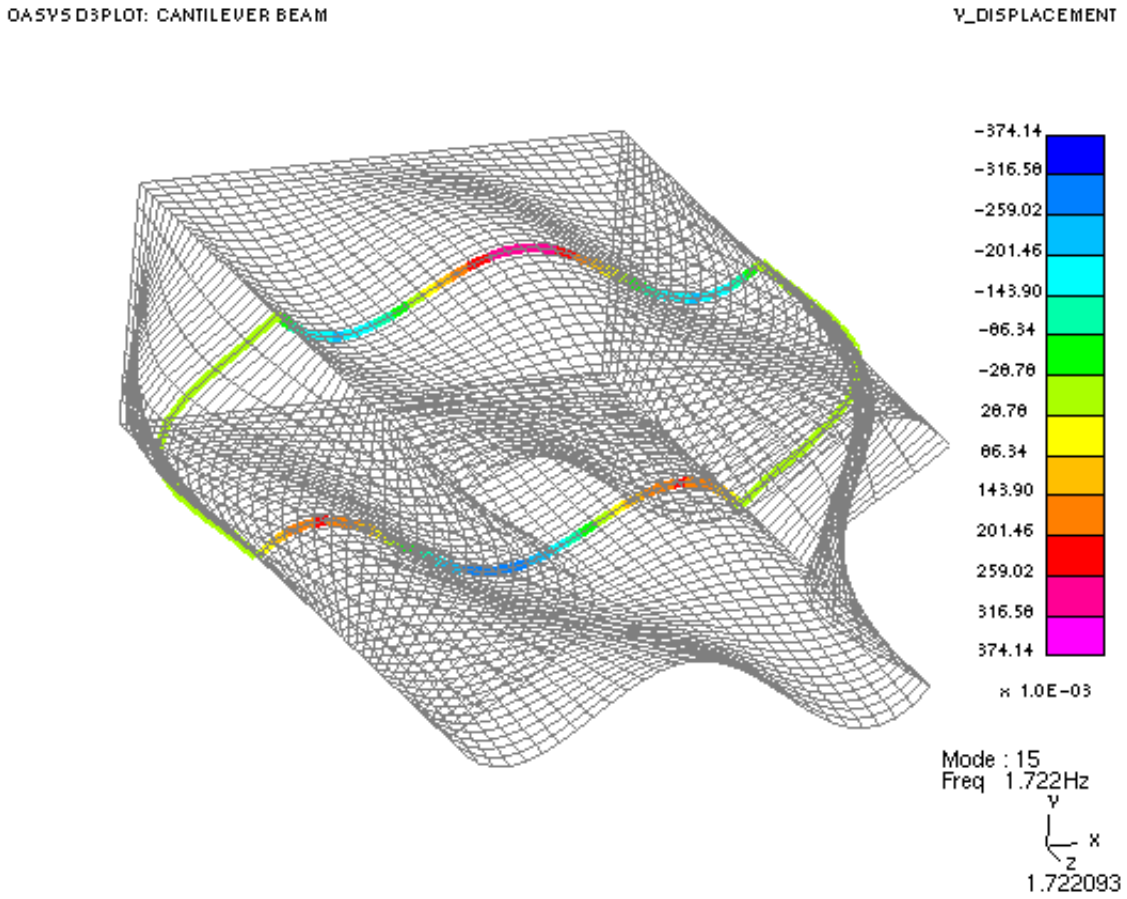
There are further commands that help the visualisation of cut-sections:



CAP 2D ELEMENTS and their **Width**

Determines whether or not 2D elements (shells and contact segments) have "caps" drawn where they are cut. A "cap" is a thick line where each 2D element is cut, and it will show the current visual properties of the element, as in the example below.

Width sets the width of these caps if drawn.



This example shows an eigenvalue analysis of a hollow box of shells. A cut section of constant Z has been applied half way along, and contours of Y displacement drawn. The "capping" of the shells shows the displacement around the cut section.

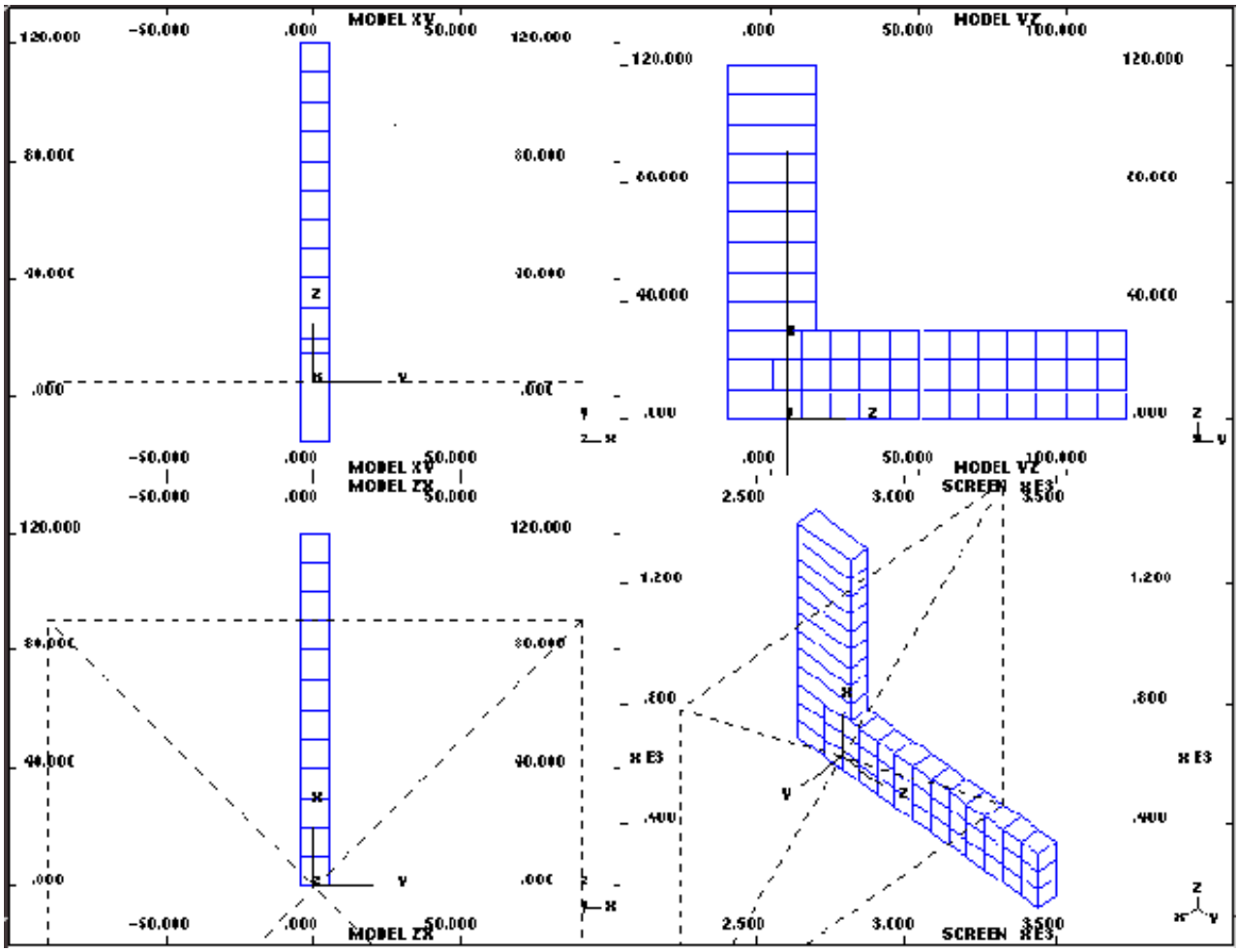
VIEW_PLANE Sets the current view to be normal to the plane.

The current view will be set so that the cutting plane X, Y, Z axes are aligned with screen X, Y, Z; giving a view exactly normal to the plane. The image is only rotated, not scaled, so it may be necessary to autoscale or apply a translation to get the view you want.

LOC_PLOT Draws a "location" plot.

This shows three standard views down each of the model X, Y and Z axes; and also the current view. The plane position is sketched on each image.

A "Graticule" is drawn on each view to provide dimensions. (Note that this image is not stored in laser files.)



6.4.9 **SAVE/RETRIEVE** Managing the storage and retrieval of cut-section definitions on disk.

There is only ever one "current" cutting-plane definition, but up to 100 such definitions can be stored in an external "**`section.cut`**" file, and any number of such files may exist.

Sections are model-independent and may be shared between dissimilar analyses.

Storing and retrieving cut-sections:

This figure shows the storage and retrieval sub-menu. The four commands in the left hand column manipulate sections as follows:

STORE	Stores the current section definition in the file.
GET	Reads a stored definition which overwrites the current one.
RENAME	Renames a stored definition.
DELETE	Deletes a stored definition.
FILE...	Lets you enter a new "**`section.cut`**" filename: Any filename is permissable, but **`section.cut`** is assumed, and the extension "**`.cut`**" is recommended (but not mandatory). Note that **`section.cut`** files are binary, and are not normally transferrable between different machine types. Nor will you be able to read or edit them. However transfers between typical workstations (using IEEE format) will usually work OK.

Only **GET** affects the current definition, the other commands leave it unchanged.

All storage and retrieval operations take place using the current "**`section.cut`**" file. If such a file has not been opened explicitly a file called section.cut is opened automatically (and an empty file of this name is created if it doesn't already exist.)

You will note that sections are stored with names as well as numbers. These are optional, but help when identifying which section does what. A maximum of 40 characters is permitted for each section name.

6.4.10 Using cut sections under OpenGL in 3D mode.

In OpenGL cut-sections are computed in a mixture of software and hardware as follows:

- OpenGL calculates in hardware the intersection between a cutting plane and the object, thus it will slice through normal objects correctly.
- However it does not compute "capping" polygons for the cut faces through 3D elements (solids and thick shells), or "caps" for 2D elements, these have to be synthesised in software and added to the plot.

While OpenGL will calculate (in hardware) the intersection between a cutting-plane and the object this imposes the limitation that the plane must be flat. Therefore Eulerian cutting, (in model space and screen space), will work in 3D mode; but Lagrangrian cutting, (in basic space, which can produce a non-planar surface), will not. The actual cut elements will be shown correctly in basic space, but if +ve or -ve sides are displayed they will not currently meet the cut elements correctly.

Therefore if you are running under OpenGL in 3D mode you will need to swap back to 2D mode (**View Box**, **M2D**) if:

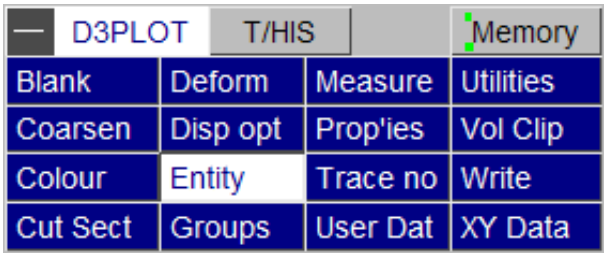
- You need to cut your structure in **Basic** space

You are warned about this if you attempt to invoke either capability while in 3D mode.

Switching to 2D mode will not impose a major speed penalty in this context, since using cut-sections drastically reduces the number of elements to be displayed.

Next Section

6.5 **ENTITY** Switching the display of entity categories on/off.

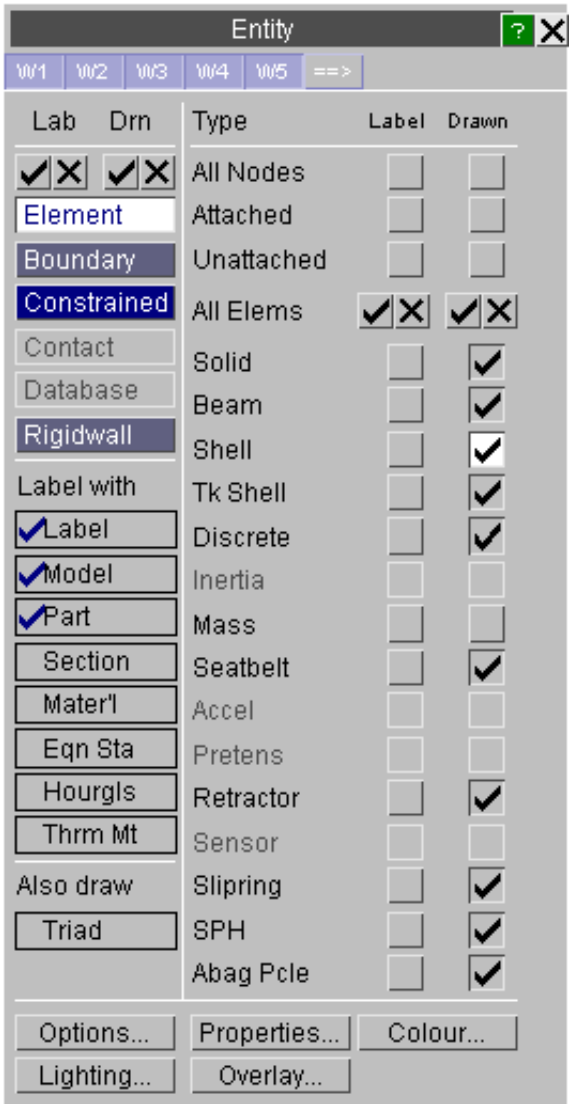


You can use **BLANKING** to control the visibility of selected nodes and elements, but sometimes you may wish to remove a complete category of entity type from the display list. The **ENTITY** menu provides this capability: when an entity category is switched off it is never displayed in subsequent plots (of any type) until explicitly turned back on again. The entity panel also controls label display. By default only the elements in a model are drawn, with no labels, node symbols or other information appended to them.

You can add extra information to plots, control the display of classes of information and label items dynamically on the screen using the menu. This can be acessed in 3 ways, the shortcut key E, the top bar menu **DISPLAY** > **ENTITIES** or the button **ENT**. This panel controls the display of elements and nodes, (ie basic "structural" items); also their symbols, labels and local direction triads as well as the display of "other" items, such as constraints, contacts, rigidwalls, etc; and also their labels, symbols and other related displayable data.

It must be stressed that these commands only permit or deny the display of *classes* of information, they do not control the visibility of individual items or models.

For example they might be used to enable the display of nodes. This would permit nodes in any models to be displayed provided they were not made invisible by some other command.



The left hand column of the panel dictates the display of the right hand column. At any one time a "master" category will be selected from the left-hand column (in this example **Elements** is selected). The "master" categories each contain further "child" categories below them. The right hand column displays the appropriate "child" categories for the selcted "master".

The **Label** columns control whether or not the items will be labelled (with the information selected under **Label with.** The **Drawn** columns control whether or not the items will be drawn. "Child"categories can be controlled individually (in the example shown the display of beams has been turned off), however they cannot be set so as to be drawn or labelled if the "master" category is not set to be (setting a "child" to be on will automatically switch the "master" setting on).

Label with determines what is actually drawn as a "label" when labelling is selected for an element or node class.

Selecting multiple labelling categories will lead to compound labels being generated (eg **M1/H1001/P12/**) and plots will become very cluttered if too much information is displayed.

As this example illustrates different models may contain different entity types. Where all currently selected models contain a type (for example Shells here) the selection box has a white background, whereas if only a subset do then the box will have a grey background to denote its "mixed" status.

6.5.1 Elements and nodes (Structural Items).

Nodes are a special case.
The display of nodes in D3PLOT is treated differently to that of elements: by default they are not drawn, but they are still "there" on a plot for the purposes of labelling and screen-picking.

This situation arises because it is convenient to be able to screen-pick and label nodes, but not useful to draw their symbols: that would just slow down graphics and clutter up the screen. Therefore the treatment of nodes is as follows:

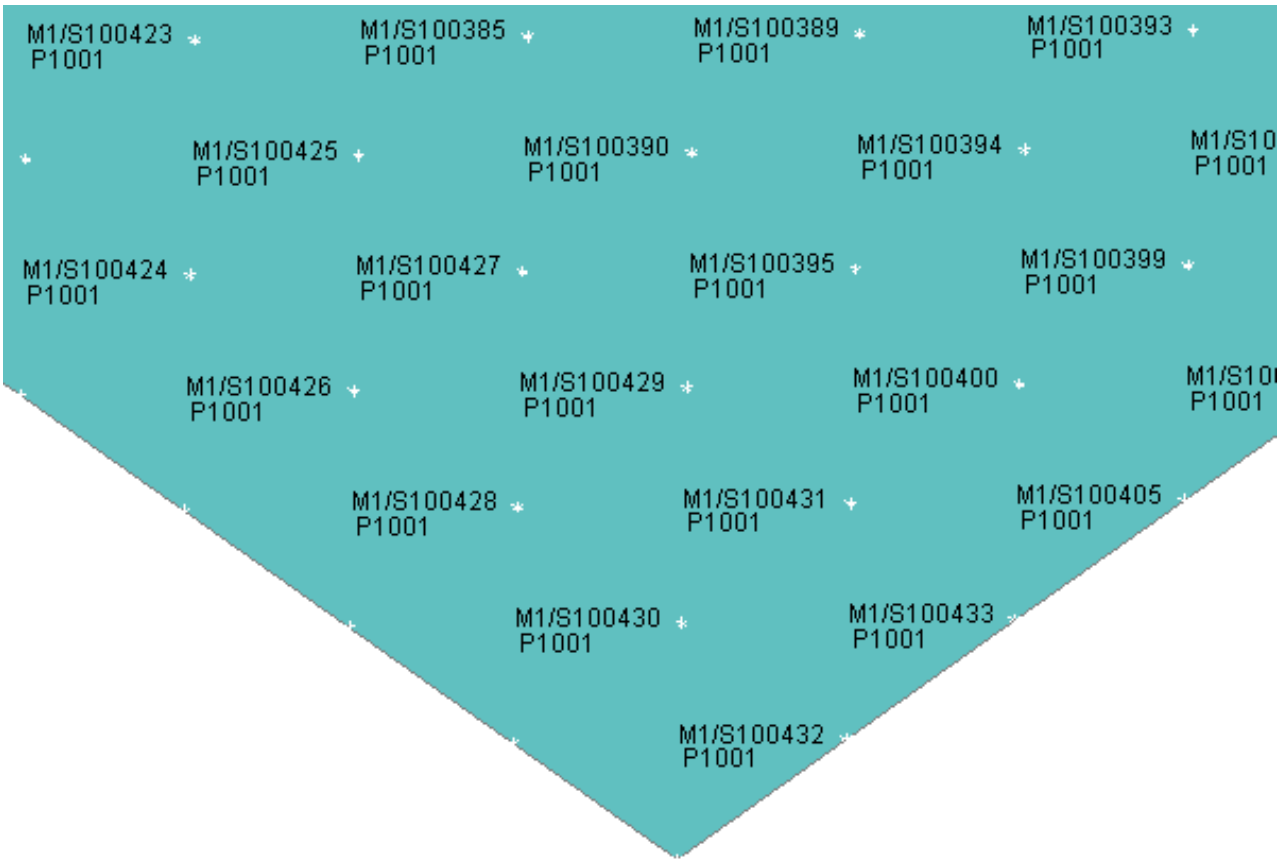
- Nodes on visible elements are always "there", even if not drawn explicitly, and may be screen-picked and labelled (by either method) at will.
- Turning the **Attached nodes** switch here **on** will cause those nodes "attached" to visible elements only to be displayed using "star" symbols.
- Turning the **All nodes** switch here **on** will cause all nodes, whether attached to an element (visible or otherwise) or not, to be displayed

Each class of element may be selected for display and/or labelling individually. Here the display of beams has been turned off.
The **ALL ELEMS** row selects them all.

Associated data: Local direction **TRIADS** are drawn for element types with coordinate systems.

6.5.2 How labelling on plots is handled for nodes and elements

Static labels are handled via the Entity panel (as described in this section). "Dynamic" labels may be added using Quick-Pick (see section 3.5)The default label is a node or element number, but a variable amount of information can be generated to form a "label" which can run to multiple lines, as this example shows.



This figure shows an example of shells which have been labelled with:

MODEL **Mnnn** for ***M***odel number <nnn>

LABEL **Snnn** for ***S***hell <nnn>.

PART **Pnnn** for ***P***art <nnn>.

D3PLOT attempts to group labels logically and to locate them so that they don’t overlap, but if you try to add too much information you will end up with a total mess on the page.
The "attached" nodes in this figure have also been switched on: these are drawn as asterisks (*) at the relevant element vertices.

Res/Constraints Displaying nodal Restraints and Constraints

If a **ZTF** file is present, then restraint and constraint data will be available for any combination of nodes on:

***BOUNDARY**	**_SPC**
	_PRESCRIBED_MOTION
	_CYCLIC
	_SLIDING
***CONTACT**	**_CONSTRAINED_XXX**
	_TIED_XXX
***CONSTRAINED**	**_LINEAR**
	_WELD_XXX
	_NODE_SET
	_NODAL_RIGID_BODY
***PART**	which is rigid

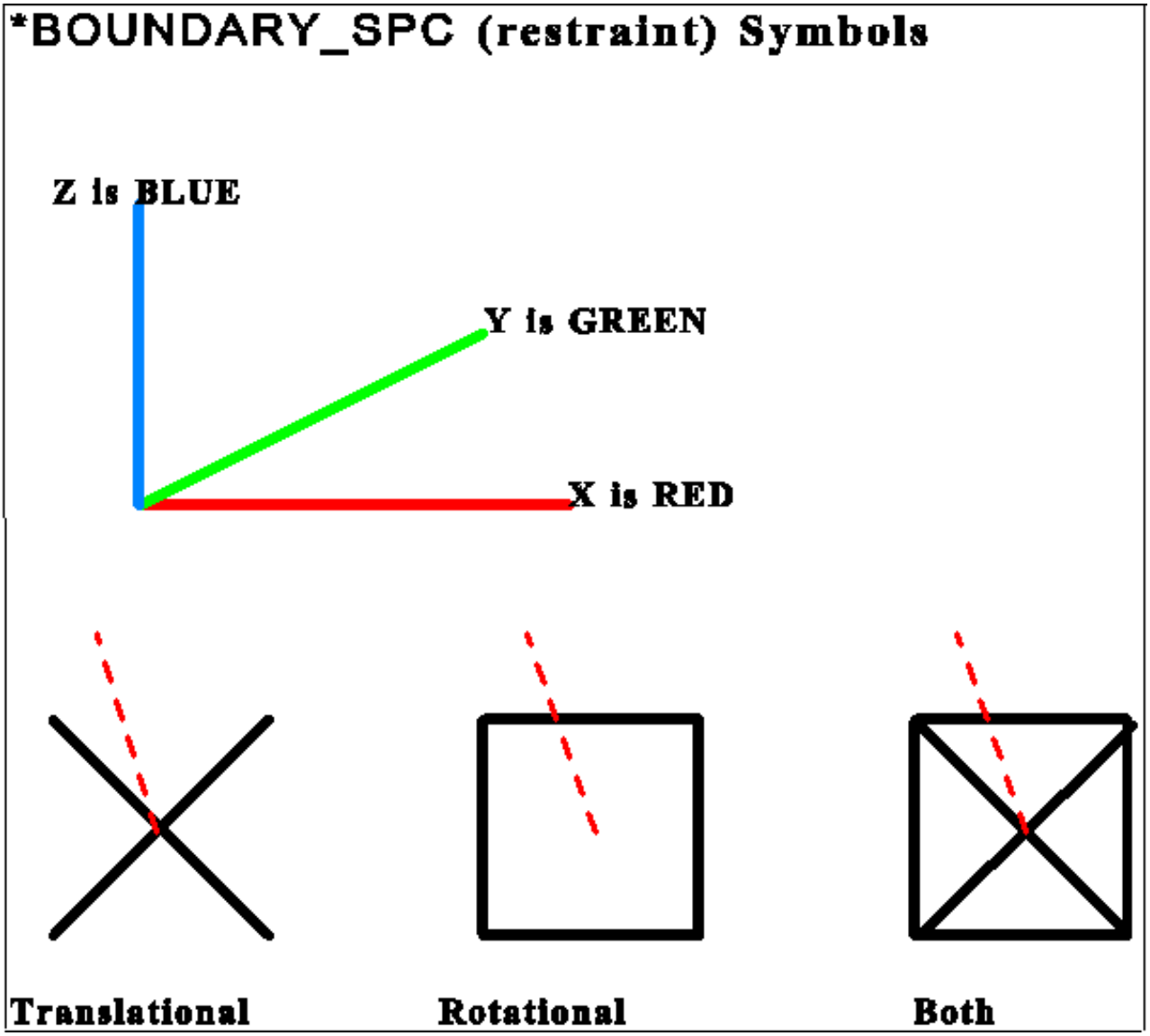
The image below shows how symbols for translational and rotational restraints and constraints are displayed.

X direction : Red
Y direction : Green
Z direction : Blue

Translational restraints have a ***cross*** at their end
Rotational restraints have a ***box*** at their end.

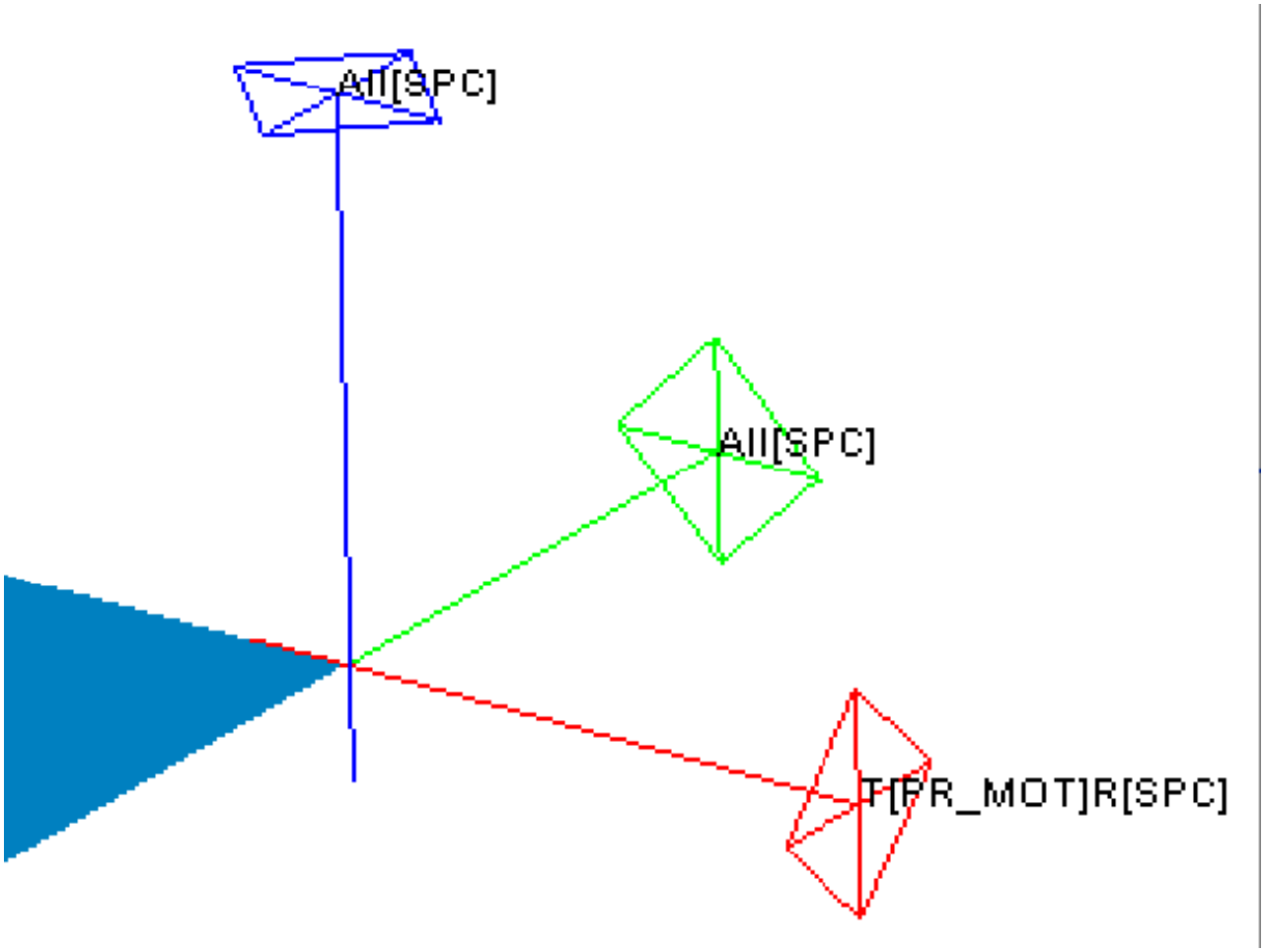
Turning on "labels" for the relevant item will give a brief description of the source of the re/constraint.

"T" designates translational restraint
"R" designates rotational restraint
"All" designates both translational and rotational



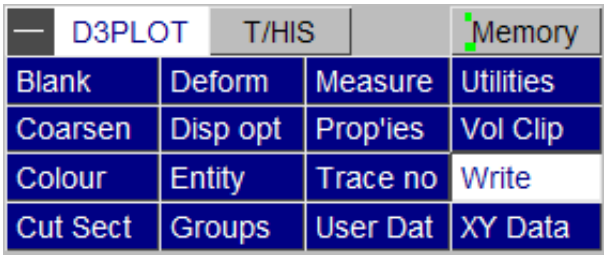
Here is an example of SPCs and Prescribed motion re/constraints from an actual model.

This image has "labels" turned on, and has been enlarged to show the symbols more clearly. Note that in the X direction translational constraint is due to a Boundary_Prescribed_Motion, whereas rotational restraint has been applied via a Boundary_SPC.



Next section

6.7 **WRITE** Listing numerical data to screen and/or file.



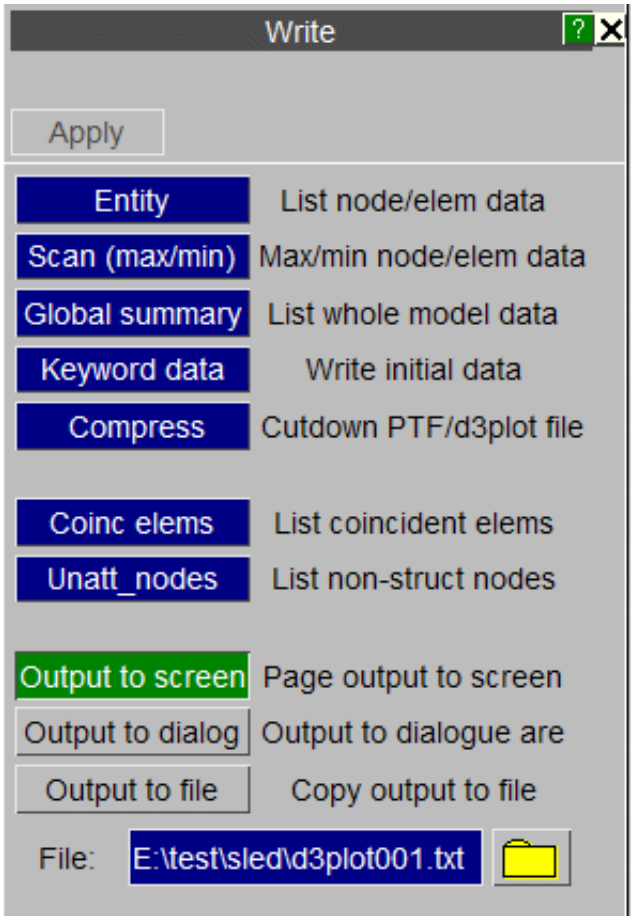
The **WRITE** command allows you to write to the screen and/or file just about any piece of information in the database files accessed by D3PLOT. It is designed to list information at a given time, if you want to plot data (in XY graphical form) you should use the **XY_DATA** command described in Section 6.8 instead.

The main **WRITE** control panel is shown right.

The **ENTITY, SCAN, GLOBAL SUMMARY, KEYWORD DATA, COMPRESS, COINC ELEMENTS** and **UNATT_NODES** commands generate output, as described in Sections 6.7.1 to 6.7.7 below.

The **OUTPUT_TO...** options define where the results are to be written. See Section 6.7.0 below.

File: is the name of the disk output file.

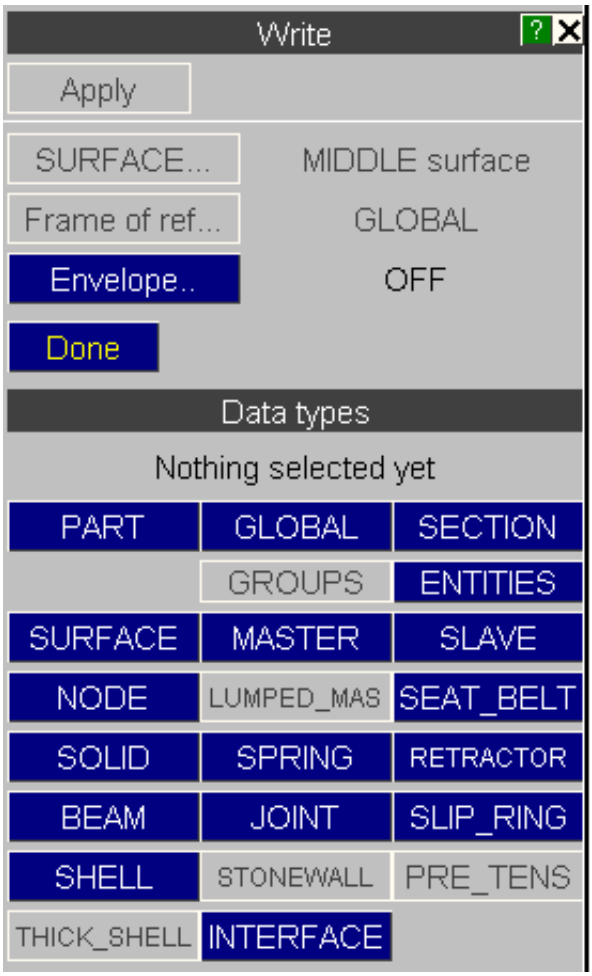


6.7.0 **OUTPUT_TO...** Choosing output destination(s) for **WRITE** output

OUTPUT_TO_SCREEN	Is a screen "listing box". This is the default since any number of lines of output can be handled by "paging".
OUTPUT_TO_DIALOGUE	Is the dialogue box. Listings up to 200 lines long can be handled by scrolling, but output longer than this will be lost. It has the advantage that is remains visible for reference during ensuing operations, whereas "listing box" output is lost.
OUTPUT_TO_FILE	Is an ASCII disk file. Output of any length may be written.

Any permutation of these options may be used at the same time, and switched on/off at will.

6.7.1 [**WRITE**] ENTITY



The figure (right) shows the **ENTITY** control panel.

"Entity" output means results for a <list> of standard entity types, in this example 90 solid elements have been chosen, at the current time.

It is also possible to write "global" (ie whole model), "material" and "surface" related information: see Section 6.7.1.7 below.

To use this panel:
- Select an entity type (eg **NODE**)
- Define a <list> of entities
- Define the data component etc
- Press **APPLY** to generate output

6.7.1.0 What does “entity” output produce?

You can write out just about any piece of information about any entity in your database. Two examples, shell and nodal output, are given below:

```
D3PLOT "write" output file.
(22-Nov-95 17:45:52)

Database file:
/users/dyna70/test/tall

Title: test1

++++++++ Data at time .35909E-01
++++++++

SHELLs: listing of UNAVERAGED
VON_MISES_STRESS (Top surf)
```

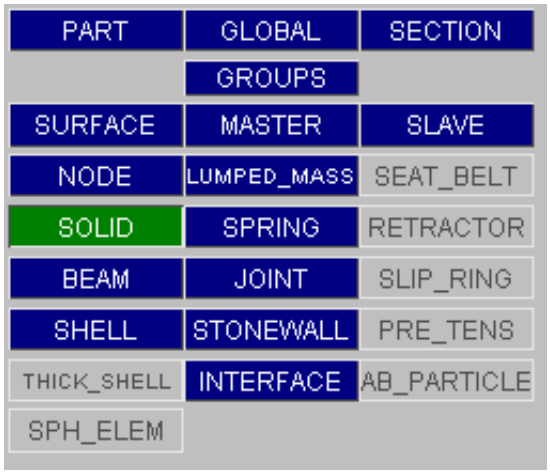
Element No:	Data value
S1	9.656122E+07
S2	1.965423E+08
S3	1.445315E+08
S4	1.689751E+08
S5	1.331927E+08
S6	1.176105E+08
S7	1.465435E+08
S8	1.561687E+08
S9	1.550533E+08
S10	1.043070E+08
S11	1.613382E+08
S12	2.154273E+08

++++++++ Data at time .35909E-01 ++++++++

Nodes: listing of VV_VELOCITY_VECTOR

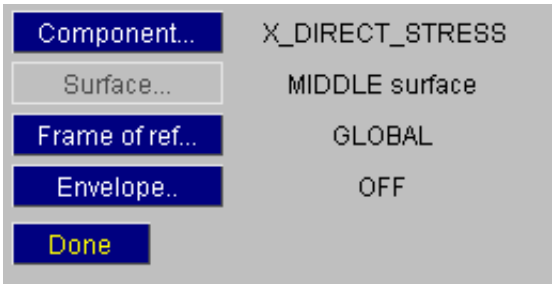
Node	VX	VY	VZ	Mag
1000	.0000E+00	.0000E+00	.0000E+00	.0000E+00
1001	-3.6481E-07	-1.5074E-07	-1.8178E-07	4.3457E-07
1002	-7.4297E-07	-1.5094E-06	-1.0158E-07	1.6854E-06
1003	-7.5988E-07	-2.2915E-06	-2.4309E-08	2.4143E-06
1004	.0000E+00	.0000E+00	.0000E+00	.0000E+00
1005	-3.2251E-07	-3.7708E-07	-1.7842E-17	4.9619E-07
1006	-7.7683E-07	-1.5576E-06	-2.8229E-13	1.7405E-06
1007	-8.3489E-07	-2.3889E-06	-2.7706E-13	2.5306E-06
1008	.0000E+00	.0000E+00	.0000E+00	.0000E+00
1009	-3.6481E-07	-1.5074E-07	1.8178E-07	4.3457E-07
1010	-7.4297E-07	-1.5094E-06	1.0158E-07	1.6854E-06
1011	-7.5988E-07	-2.2915E-06	2.4308E-08	2.4143E-06
2000	.0000E+00	.0000E+00	.0000E+00	.0000E+00
2001	3.6482E-07	-2.2947E-07	1.8178E-07	4.6775E-07
2002	7.4297E-07	-1.4183E-06	1.0158E-07	1.6043E-06
2003	7.5988E-07	-2.4126E-06	2.4308E-08	2.5295E-06
2004	.0000E+00	.0000E+00	.0000E+00	.0000E+00

6.7.1.1 Selecting an entity type

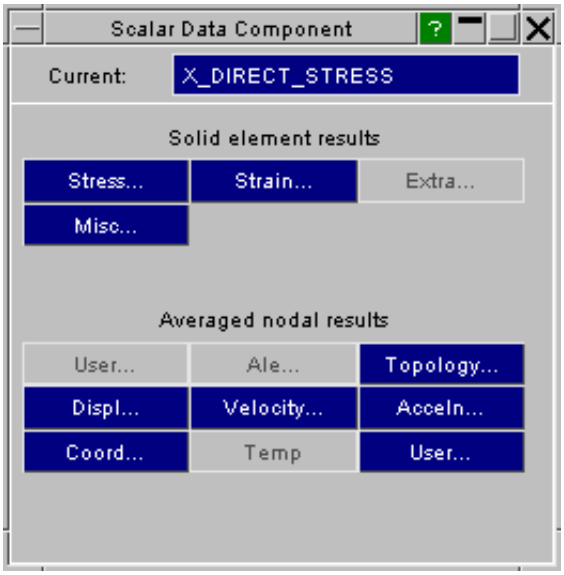


Firstly choose the type of entity for which you want to extract data.

In this example the user has chosen **Solids**. It will then become possible to select the data component, shell surface and frame of reference. These will default to what was most recently plotted in the first window of the current model.



6.7.1.2 **COMPONENT** Choosing a data component.



Every entity type has at least one data component associated with it, and most have several.

For example the user above has chosen **SOLID** elements, so the components shown will be those available for solids, and also nodal data that can be averaged at nodes for solids.

It is also possible to **WRITE** "non-data" components such as element topology and part number.

6.7.1.3 **SURFACE** Shell surface selection

If Shell elements are selected, the **SURFACE** button is used to determine for which surface the results will be written.

You are referred to Section 12.8.2.2 for a more detailed description of shell surfaces.

Note 1: You have to define surfaces for thick shell output in **WRITE**. This is not necessary when plotting since it is possible to draw all three sets of values simultaneously.

Note 2: Top and bottom shell surfaces are not the element "outer fibres" if the default Gaussian integration scheme is used. See Section 12.8.2.2 for more details.

Note 3: It is possible to make LS-DYNA generate output at other than the three surfaces described here, in which case "layers" will be available for selection as well as surfaces. Typically this will be used for a composites analyses: see Section 12.8.2.2.

6.7.1.4 **FRAME_OF_REF** Local, global or cylindrical

When a directional stress or strain component is chosen for solids, shells or thick shells the frame of reference for output must be defined.

This defaults to global, but you can opt for element local axis system output.
See Section 12 for more information.

Note: It is possible, but unusual, to make LS-DYNA write stresses for some material models in the material axis system. This would usually only be used for the output of orthotropic material results and laminates. See Section 12.8.2.3 for more details.

There is no way to tell from the database file that directional stresses for these element types have been written this way, and D3PLOT assumes that they are in the global system: **it is your responsibility to interpret your results correctly.**

6.7.1.5 **ENVELOPE**

Instead of reporting results at a single time-state, D3PLOT can calculate the maximum or minimum values occurring over a selection of time-states.

Once a component has been selected the user can press **APPLY** and the output will be generated. For the example above it will look like this (taken from file output):

```
D3PLOT "write" output file. (22-Nov-95 12:27:49)
Database file: /users/dyna70/test/tall
Title:         test1
++++++++ Data at time .35909E-01 ++++++++
SOLIDs: listing of UNAVERAGED GLOBAL X_DIRECT_STRESS
Element No:     Data value
--------------------------------
        H1     5.155513E+06
        H2    -1.659268E+07
        H3    -1.020761E+07
        H4    -2.168100E+07
        H5    -2.182096E+07
        H6    -1.125487E+07
        H7     5.599310E+05
```

and so on to the 80th element

Scalar and compound data components

In the example above the component **X_DIRECT_STRESS** was used, this is a "scalar" value that generates only one number per element. In **WRITE** it is sometimes convenient to be able to be able to write out "compound" data, ie more than one number per entity. Examples of this are stress and strain tensors for elements, force and moment summaries for beams, etc.

Therefore you will find such components available in the "specific" component panels, generally at the bottom of component lists.

For example under stress there are **ST_STRESS_TENSOR** and **PS_PRINC_ STRESS_SUMMARY** components.

Examples of output from these is given below:

++++++++ Data at time .35909E-01 ++++++++

SOLIDs: listing of GLOBAL ST_STRESS_TENSOR

El No:	Sx	Sy	Sz	Txy	Tyz	Tzx
H1	5.1555E+06	-1.1453E+07	1.7028E+06	2.8189E+07	-9.2431E+06	5.0026E+06
H2	-1.6593E+07	-1.9981E+07	-1.2872E+07	3.5470E+07	-4.0756E+07	1.1448E+07
H3	-1.0208E+07	1.3946E+07	-9.4400E+06	3.2251E+07	-2.9166E+07	6.6445E+06
H4	-2.1681E+07	2.9476E+07	-2.8595E+07	1.9464E+07	-3.1784E+07	7.4274E+05
H5	-2.1821E+07	3.9145E+07	-3.0982E+07	-2.4756E+06	-3.4087E+07	-1.5587E+06

++++++++ Data at time .35909E-01 ++++++++

SOLIDs: listing of PS_PRINC_STRESS_SUMMARY

Principal stresses			Cosines		
El No:		cx	cy	cz	
H1	Maximum :	2.6341E+07	.79188	.60700	-.06694
	Middle :	4.5134E+06	.21564	-.17539	.96059
	Minimum :	-3.5449E+07	.57134	-.77510	-.26978
H2	Maximum :	3.1369E+07	.40057	. 72321	-.56259
	Middle :	-3.7253E+06	.76642	.07203	.63829
	Minimum :	-7.7090E+07	.50214	-.68686	-.52543

"Geometric" and other "miscellaneous" data components

As well as analytical data values it is possible to extract "geometric" data components. For nodes and elements these appear under the **TOPOLOGY** and **MISC...** generic buttons.

Here is an example of component **LN_LIST_OF_NODES** output for solids:

```
  ++++++++ Data at time .35909E-01 ++++++++
SOLIDs: listing of LN_LIST_OF_NODES
El No:   n1    n2    n3    n4    n5    n6    n7    n8
-------------------------------------------------------
H1       1     11    12    2     101   111   112   102
H2       2     12    13    3     102   112   113   103
H3       3     13    14    4     103   113   114   104
```

For element types for which no analytical data are available (lumped-masses, springs, joints, seat-belt types) only topologically derived components are available. For example here is a typical **SUMMARY** output for springs:

```
          ++++++++ Data at time .35909E-01 ++++++++
SPRINGs: listing of SUMMARY
Spring No:     Material        Type        N1     N2
-----------------------------------------------------
SP1                 1     Trans spring   243    101
SP2                 1     Trans spring   151    233
SP3                 1     Trans spring   252    110
SP4                 1     Trans spring   160    242
```

For a full list of data components available in the **WRITE** command see Sections 12.3 - 12.14.

6.7.1.6 Averaged and Unaveraged element results written at nodes

Element data written at nodes has to be averaged. All elements attached to that node which could legitimately contribute the required data component are included, and the unweighted average is used. If elements are blanked or volume-clipped they are only excluded if the **MATERIAL_IGNORED** or **CLIPPING_IGNORED s**witches respectively are on.

Problems can arise in the following situations:

On material boundaries:	For the node there is no "parent" element to give a definitive material number, as there is when plotting, so averaging will always take place across dissimilar materials.
At element type junctions:	For the node there is no "parent" element type. So where, for example, a node is common to a solid and a shell, the results will be averaged across the two types.

Therefore you should take care when using **WRITE** to extract element data averaged at nodes.

Whenever **WRITE** tabulates data it prefaces it with its full data component name and, for nodal results, it also states whether the figures are "averaged" or "unaveraged".

6.7.1.7 Special "entity" types.

As well as the standard node and element categories, familiar in other contexts, you can select the following "global" and other information for output:

GLOBAL	Refers to the whole model. LS-DYNA computes energies, rigid body velocities and mass; all of which may be output. See Section 12.3 for more details.
PART	Refers to solid, shell, beam and thick-shell (but not spring) **PART**s. LS-DYNA calculates energies, velocities and mass for these; all of which may be output. See Section 12.4 for more details.
SURFACE	Refers to contact surfaces. LS-DYNA writes nodal forces to the **.CTF** file, and D3PLOT can sum these to produce overall forces on a surface. (You can also choose to compute **MASTER** or **SLAVE** side forces only.) See Section 12.5 for more details.
AIRBAG	Refers to "airbags" - effectively control volumes - used with the Airbag Particle method. This capability is still under development in LS-DYNA, and only a limited amount of information is available for post-processing, see Section 12.13 for more information.

Output for these categories of information under **ENTITY** is selective: you can choose those materials or surfaces for which you want to see results.

To list complete summary information you can use the **GLOBAL_SUMMARY** option in the main **WRITE** menu instead: this summarises data for all materials or all contact surfaces, which may be more convenient.

Next section

6.7.2 [**WRITE**] **SCAN**

This figure (right) shows the **SCAN c**ontrol panel.

"Scan" output means the maximum and minimum results for a given entity/data component combination.

In this example the user is **SCAN**ning for the max/min Global X stress (middle surface) in shells.

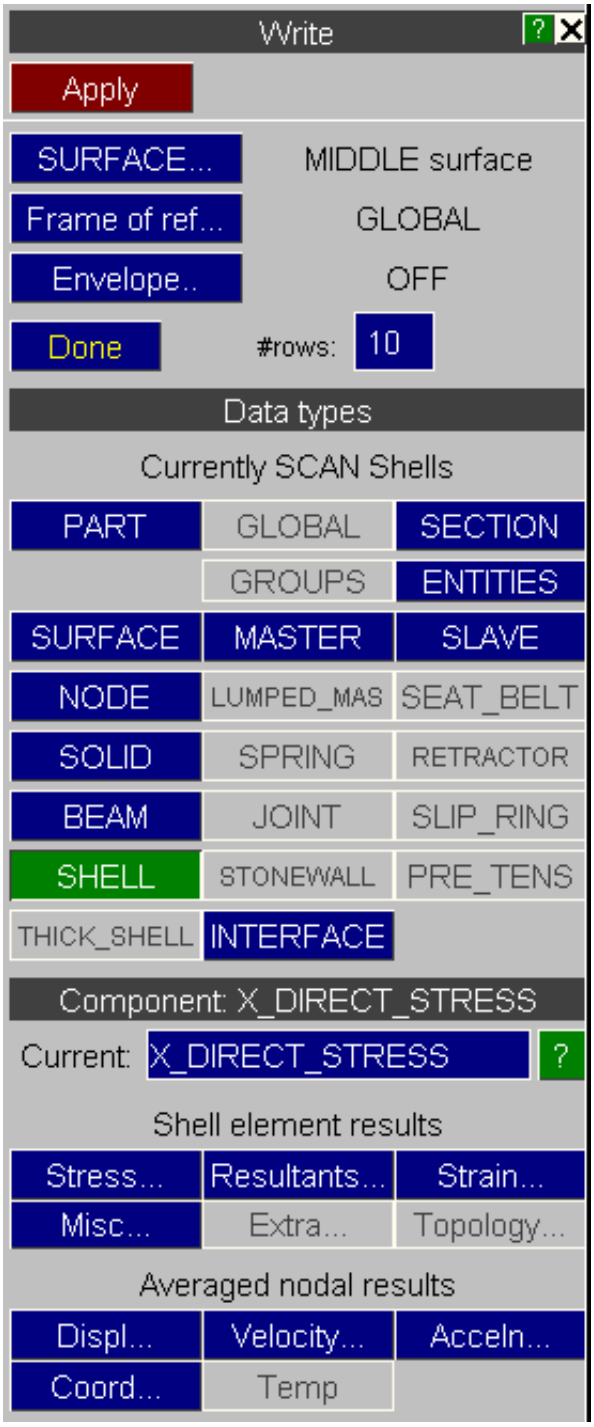
To use this panel:

- Select an entity type (eg **NODE**)

- Define the data component etc

- Define **#rows** to calculate

- Press **APPLY** to generate output



6.7.2.0 What does a "scan" produce?

A "scan" operation searches the list of entities defined, and outputs the <#rows> maximum and minimum scalar values that are eligible for plotting (see Section 6.7.2.6 below) in a table. An example of typical output is shown below:

D3PLOT "write" output file. (22-Jan-96 16:59:01)

```
Database file:  /users/dyna70/test/tall
Title:          test1
```

++++++++ Data at time .35909E-01 ++++++++

SHELLs: scan of 10 max/min UNAVERAGED
GLOBAL X_DIRECT_STRESS (Mid surf)

	Greatest values		Smallest values	
NumElem		Value	Elem	Value
1	S1	2.29090E+06	S15	-7.78900E+07
2	S5	1.71183E+06	S14	-7.57228E+07
3	S18	-6.71435E+06	S13	-6.80775E+07
4	S9	-8.15491E+06	S16	-5.27643E+07
5	S10	-8.27366E+06	S12	-3.27425E+07
6	S17	-1.30454E+07	S3	-3.12505E+07
7	S4	-1.65524E+07	S11	-2.84226E+07
8	S8	-1.70622E+07	S6	-2.66353E+07
9	S7	-2.12947E+07	S2	-2.16315E+07
10	S2	-2.16315E+07	S7	-2.12947E+07

6.7.2.1 Selecting an entity type.

This is done in exactly the same way as for **[WRITE] ENTITY** output, and exactly the same range of entity types is available. (See Section 6.7.1.1)

However, in this case, no <list> is selected. Instead all entities of the chosen type which are eligible for plotting (see Section 6.7.2.6 below) are included in the scan operation.

6.7.2.2 **COMPONENT** Selecting a data component

This is done in exactly the same way as for **[WRITE] ENTITY** output, see Section 6.7.1.2, but only scalar data components are permitted.

6.7.2.3 **SURFACE** Selecting a shell surface

This is done in exactly the same way as for **[WRITE] ENTITY** output, see Section 6.7.1.3.

6.7.2.4 **FRAME_OF_REF** Selecting global/local/cylindrical frame of reference

This is done in exactly the same way as for **[WRITE] ENTITY** output, see Section 6.7.1.4.

6.7.2.5 **#rows** Selecting the number of rows of output

By default 10 rows of output are generated, but any number up to a maximum of 200 is permitted. This controls the number of #maximum and #minimum values that are stored and output.

6.7.2.6 What does "eligible for plotting" mean?

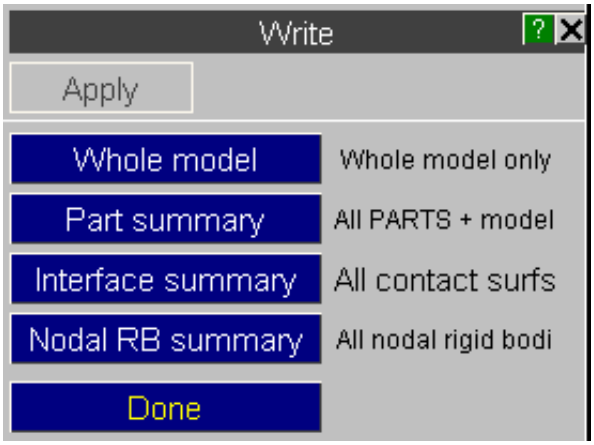
Entities are only included in a "scan" operation if they are currently available for plotting. This permits you to restrict the range of entities scanned using the same tools that are used to restrict what is plotted. The things that exclude them are:

Blanking: Blanked nodes/elements are excluded. The contribution of blanked elements to averaging data at nodes depends on the **BLANKING_IGNORED** switch.

Vol clipping: Volume-clipped nodes/elements are excluded. The contribution of clipped elements to averaging data at nodes depends on the **CLIPPING_IGNORED** switch.

Note that an element which is off the current screen because of scale or view-point is still "eligible" for plotting.

6.7.3 [WRITE] GLOBAL_SUMMARY



This figure shows the Global summary control panel.

This controls the display of the summaries of "global" (whole model, material and contact surface) data.

This data can also be extracted in a more selective fashion using the **ENTITY** command.

6.7.3.1 WHOLE_MODEL Summary data about the whole model only

This is a summary of the energies and rigid-body velocities of the whole model:

GLOBAL data: Summary of whole model data

	Energies x 1.0E+12			Velocities x 1.0E+00			
Matl:	**Kinetic**	**Internal**	**Total**	**Vx**	**Vy**	**Vz**	**Mass**
Sum	**2.092**	**0.130**	**2.222**	**2.572**	**14.320**	**584.463**	**n/a**

6.7.3.2 PART_SUMMARY Summary data for all PARTS + whole model

This is the same as above, but broken down by material.

GLOBAL data: Summary of data by PART

	Energies x 1.0E+12			Velocities x 1.0E+00			
Matl:	**Kinetic**	**Internal**	**Total**	**Vx**	**Vy**	**Vz**	**Mass**
1	**0.001**	**0.018**	**0.019**	**9.772**	**104.088**	**-87.916**	**1.152E+06**
2	**0.001**	**0.001**	**0.002**	**16.120**	**-19.311**	**-37.918**	**7.879E+06**
3	**0.055**	**0.106**	**0.161**	**-5.964**	**100.652**	**-844.292**	**2.099E+06**
4	**0.033**	**0.006**	**0.038**	**-15.872**	**26.443**	**-384.108**	**4.536E+06**
5	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**0.000**	**1.024E+06**
Sum	**0.090**	**0.130**	**0.220**	**3.220**	**17.916**	**-234.561**	**1.669E+07**
Mode	**2.092**	**0.130**	**2.222**	**2.572**	**14.320**	**584.463**	**n/a**

Note: The "Sum" of all material values is frequently less than the (whole) "Model" values above. This is because the "Model" values include the contribution from any lumped-masses, contact friction, etc that are not part of "material" data.

6.7.3.3 INTERFACE_SUMMARY Summary forces on all contact surfaces

This lists the global [x,y,z,magnitude] force vectors for all contact surfaces. The values are computed from the individual nodal forces in the **.CTF** file, and are presented separately for slave and master sides of each surface.

```
GLOBAL data: Summary of Interface data
Surface     Fx            Fy            Fz            Resultant
-------------------------------------------------------------------
1 Slave      1.26670E+08  2.61471E+09                 8.07831E+09
                                        7.64240E+09
  Master    -1.26670E+08 -2.61471E+09                 8.07831E+09
                                       -7.64241E+09
2 Slave      0.00000E+00  0.00000E+00                 0.00000E+00
                                        0.00000E+00
  Master     0.00000E+00  0.00000E+00                 0.00000E+00
                                        0.00000E+00
3 Slave      0.00000E+00  0.00000E+00                 0.00000E+00
                                        0.00000E+00
  Master    -4.27069E-02  3.57985E-02                 8.63139E+00
                                        8.63122E+00
4 Slave     -1.77204E-02  0.00000E+00                 2.35839E+02
                                        2.35839E+02
  Master     0.00000E+00  0.00000E+00                 0.00000E+00
                                        0.00000E+00
```

Note 1: Normal contact surfaces with both slave and master sides will have equal and opposite forces on each side. For example surface #1 here.

Note 2: The contact force file only contains forces for nodes on segments. Where discrete nodes form one side of a surface then no forces are written for that side. For example surface #4 above is "Discrete nodes (slave side) impacting surface (master side)", and forces only appear for the master side.

Note 3: Single-surface contacts only have a slave side, and the net force on such a contact surface is zero. However plots will show a distribution of force on the individual segments of the contact.

Note 4: Contact surface types using a one-way treatment (eg types 10 and 21) write forces for the master side only. Surface #3 above is such a surface.

Note 5: Contacts using a constraint rather than a penalty formulation do not generate forces. (However contacts using "soft constraint" still generate forces.)

6.7.3.4 NODAL_RB_SUMMARY Summary energy and velocity for nodal rigid bodies

Nodal rigid bodies write energy and rigid body results in the same way as materials:

GLOBAL data: Summary of Nodal rigid body data

	Energies x 1.0E+12			Velocities x 1.0E+03			
Matl:	Kinetic	Internal	Total	Vx	Vy	Vz	Mass
1	0.000	0.000	0.000	0.025	0.000	0.000	8.0000E-03
2	0.000	0.000	0.000	0.000	0.010	0.000	2.0000E-02
Sum	0.000	0.000	0.000	0.007	0.007	0.000	2.8000E-02
Mode	2.248	0.000	2.248	27.127	0.088	9.480	n/a

Next section

6.7.4 [WRITE] COINC_ELEMS Output of coincident element lists

A "coincident" element is one that shares the same topology as another element of the same type, but not necessarily in the same order. Coincident elements of different types, for example contact segments on shells, do not qualify.

There are no further arguments or sub-menus for this command, and an example listing is:

```
COINCIDENT ELEMENT CHECK:
=========================

SOLIDs

Element No:   Coincident element(s):

-------------------------------------

[None found]

-------------------------------------

BEAMs
Element No:Coincident element(s):
-------------------------------------
B13001     B13002   B13003     B13004      B13005       B13006      B13007
B13002     B13003   B13004     B13005      B13006       B13007
B13003     B13004   B13005     B13006      B13007
B13004     B13005   B13006     B13007
B13005     B13006   B13007
B13006     B13007
-------------------------------------
[Total of 21 coincident BEAMs]

SHELLs

Element No:   Coincident element(s):

-------------------------------------

[None found]

---------------------------------------------------------------------

INTERFACEs
Element No: Coincident element(s):
-------------------------------------
I30         I81
I102        I153
-------------------------------------
[Total of   2 coincident INTERFACEs]
```

Every element category in the file is checked through and a summary report as above generated. As in the example above **[None found]** is reported when no coincident elements are found in a particular category.

6.7.5 [WRITE] UNATT_NODES Lists of unattached (non-structural) nodes

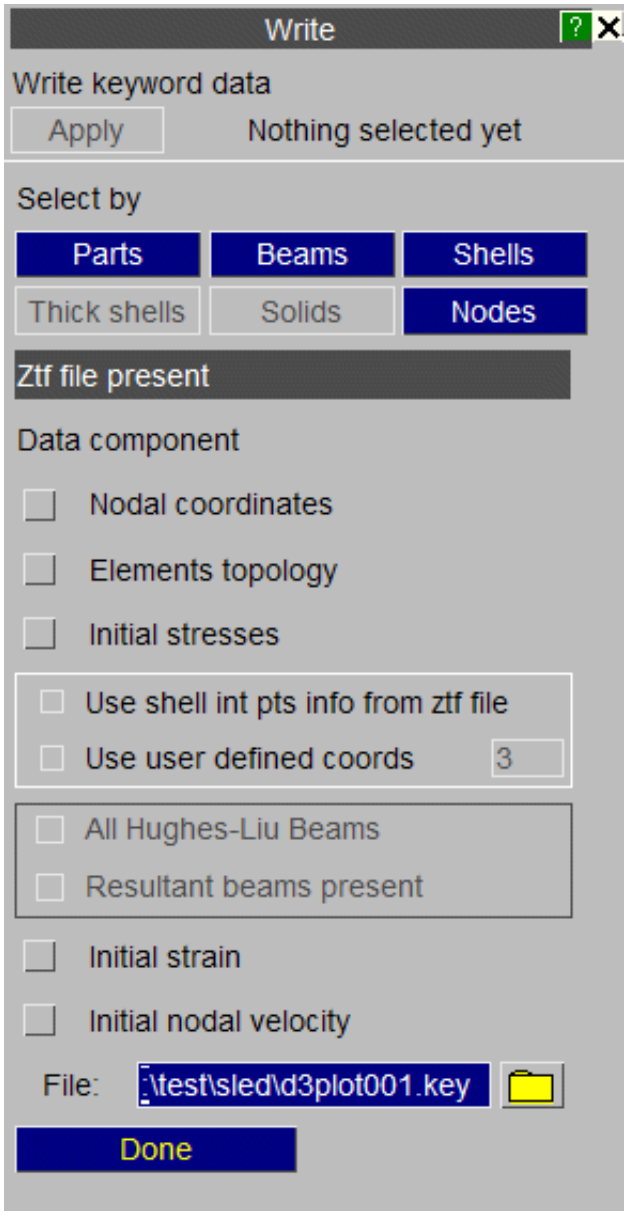
D3PLOT considers a node to be "unattached" if it is not part of the topology list of any element. It does not, for example, know about

A typical listing is:

```
Unattached nodes
--------------------------------------------------------------------------------------------------------------------------------
3007 4007 6005 10002
10008100091001010011
10018100191002010021
10502105031050510506
10511105121051410515
10521105221052310524
11004110051100611007
11012110131101411015
11020110211102211023
--------------------------------------------------------------------------------------------------------------------------------
```

Next section

6.7.6 [WRITE] KEYWORD DATA



This figure (right) shows the **KEYWORD DATA** control panel.

"keyword data" output means results for a <list> of standard entity types written into a keyword file instead of output to screen. The output is similar to that of [WRITE] ENTITY, but it has been arranged into DYNA keyword format so that it can be read into DYNA as initial data of a model. In this example 10 shell elements and 10 beam elements have been chosen for keyword Date output..

To use this panel:

- Select an entity type (eg **Beams**) or for a mixture of entity types select **Entities**
- Define a <list> of entities in the selection popup box
- Select the data component(s)
- Select the output .key file
- Press **APPLY** to generate output

What does a "keyword data" output produce?

You can write out five properties in keyword format about any entity in your database. One example with 10 shells and 10 beams selected for output is shown below.

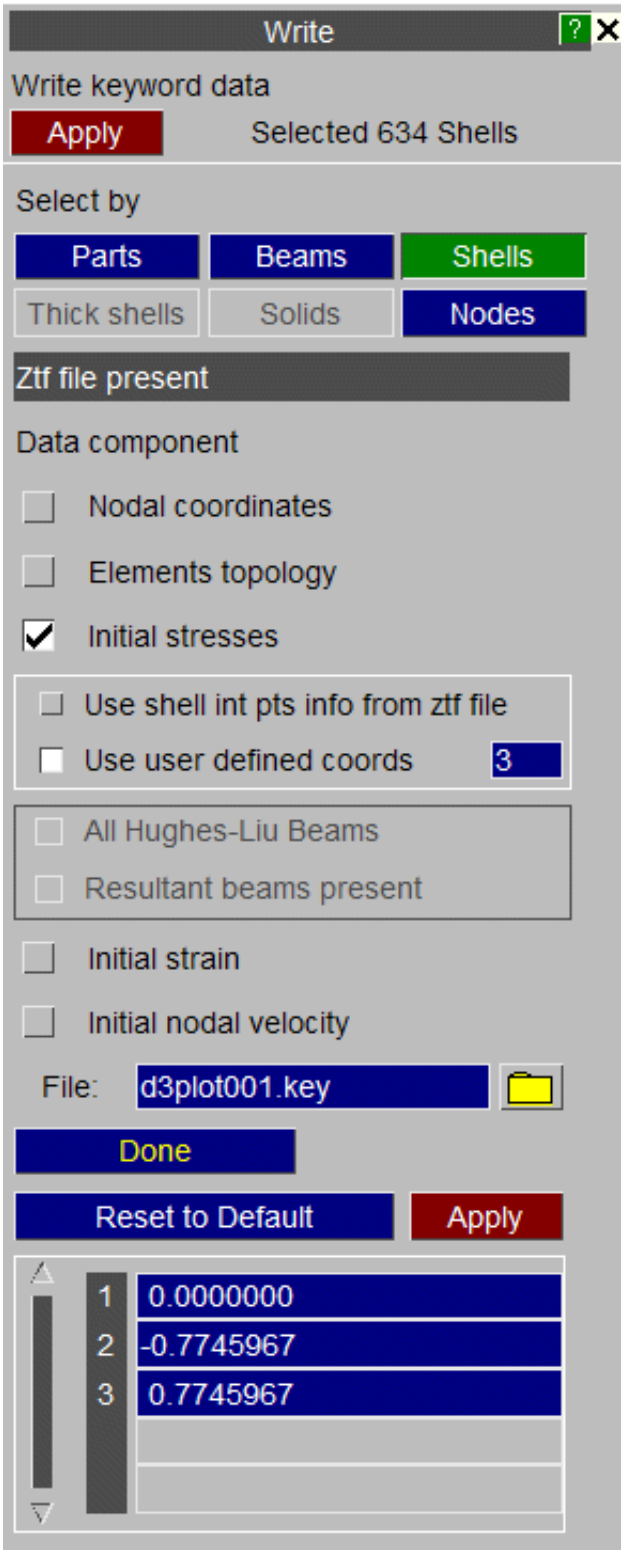
Select By

You can select the elements to write out for in Select By. If you're selecting elements with a mixture of element types then you should press "Entities". A list of all entities will be available to select in the popup menu. If you wish to write out for a single element type, then you should press the element type, i.e. Beams/Shells/Thick shells/Solids/Nodes, where a list of elements with the chosen type will be listed for selection. In our example, 10 shells and 10 beams are a mixture of element type, so Entities was pressed selection.

INITIAL STRESS and INITIAL STRAIN

Special attention is given to *INITIAL_STRESS_SHELL/TSHELL and *INITIAL_STRESS_BEAM as they both can write information about more than one integration point of the element chosen. The ztf output from Primer will include the information of integration points for any shell and beam included in the model. From the ztf file D3plot will look into *SECTION_SHELL/TSHELL, *INTEGRATION_SHELL and *PART_COMPOSITE for shells' integration point information and *INTEGRATION_BEAM and *SECTION_BEAM for beams' integration point information. Combined with the data at these integration points contained within the ptf file, we are able to write full initial keywords.

When a ztf file is not present, user defined number of integration points of shells and their coordinates can be set as shown in the figure on the right. In this case, the number of integration points is set as MAXINT from the CONTROL Card in ptf files at default, which is the number of integration points the ptf file has data for. By selecting to "Use user defined coordinates" the editing panel for the coordinates will appear at the bottom of the WRITE panel. The values in that are decided using Gaussian integration rule with the specified number of integration points. You can edit the values by Apply or Reset to default values. In the example, Gaussian rule cooordinates for 3 integration points have been edited from 0, -.7745967, +.7745967 to 0, -0.5, 0.5.



Only Hughes-Liu integration beams can have cross section integration with quadrature integration rules. With a ztf file D3plot can find this information itself so the option is always greyed out. However, without a ztf file you need to decide whether they are all Hughes-Liu beams in which case the number of integration points will be decided by the BEAMIP for beams in the Control Card in the ptf file. If "Resultant beams present" is selected, it'll assume there's only one integration point in all the beams.

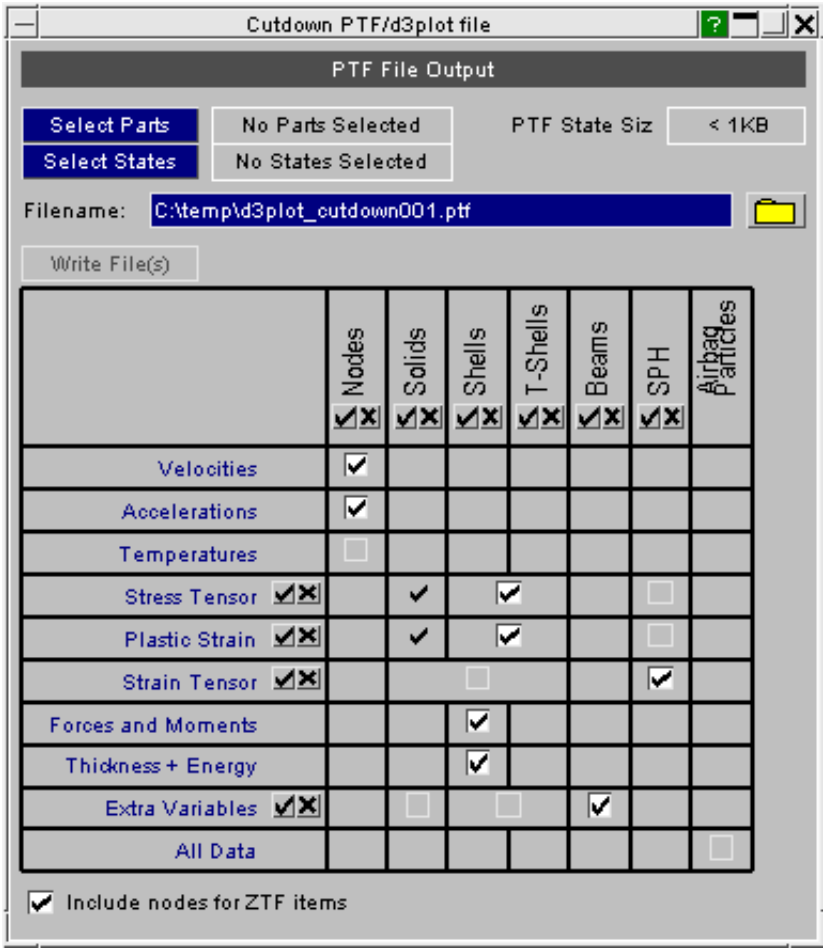
There are cases where BEAMIP and MAXINT is set zero in the model which leads to no stress information for Beams or Shells respectively, D3plot therefore will not write *INITIAL_STRESS_BEAM or *INITIAL_STRESS_SHELL.

NODAL COORDINATES and INITIAL NODAL VELOCITY

When nodal information of elements is required, such as Nodal coordinates (*NODE) and Initial nodal velocity of shell (*INITIAL_VELOCITY_NODE), beam and solids, for the elements selected, the nodes on such elements will be automatically flagged for nodel information output. Please note that by selecting all Entities, it will only include nodes on all Entities, this does not guarantee all nodes in the whole model to be included for information output.

```
*KEYWORD
$ Keyword file written by Oasys D3PLOT from plot file
$ Z:\testfiles\cantilever\beam_shell.ptf
$
$
$ ++++++++ Data at time 0.20008E-02 ++++++++
*INITIAL_STRESS_BEAM
12          2           4          1
3.594E+01  0.000E+00  0.000E+00  0.000E+00  0.000E+00  1.994E-01  0.000E+00
3.594E+01  0.000E+00  0.000E+00  0.000E+00  0.000E+00  1.994E-01  0.000E+00
-3.592E+01 0.000E+00  0.000E+00  0.000E+00  0.000E+00  2.761E-01  0.000E+00
-3.592E+01 0.000E+00  0.000E+00  0.000E+00  0.000E+00  2.761E-01  0.000E+00
*INITIAL_STRESS_SHELL
31         1           3         0         0
0.0000000  3.479E+00  -3.919E-04 4.838E-02 3.005E-07  4.622E-06 4.487E-01  0.000E+00
-0.5000000 -1.094E-02 -7.716E-07 2.783E-02 -5.075E-07 4.560E-06 1.809E-01  0.000E+00
 0.5000000 -3.501E+00 3.903E-04  7.282E-03 -1.316E-06 4.498E-06 -8.687E-02 0.000E+00
*INITIAL_STRAIN_SHELL
31
-2.756E-07  -7.238E-08  4.445E-07  -9.425E-12  8.468E-11  1.994E-01  3.360E-06
-5.005E-05  1.498E-05   1.511E-05  -2.449E-11  8.357E-11  1.994E-01  -1.613E-06
*ELEMENT_BEAM
12     8        100   101     127
*ELEMENT_SHELL
31     1        80     81     78     206
*NODE
78       8.989016E+01  1.500000E+02       3.950482E+00     0.          0.
80       9.986085E+01  1.600000E+02       4.715574E+00     0.          0.
81       8.989016E+01  1.600000E+02       3.950481E+00     0.          0.
100      9.999877E+00  2.500000E+02       4.969115E-02     0.          0.
101       1.999884E+01 2.500000E+02       1.934823E-01     0.          0.
206      9.986085E+01  1.500000E+02       4.715574E+00     0.          0.
*INITIAL_VELOCITY_NODE
78  -2.744E+02    8.881E-04   5.102E+03
80  -3.398E+02    -7.816E-05  5.955E+03    0.000E+00   0.000E+00   0.000E+00
81  -2.744E+02    -9.552E-04  5.102E+03    0.000E+00   0.000E+00   0.000E+00
100 -2.869E-01    0.000E+00   5.765E+01    0.000E+00   0.000E+00   0.000E+00
101 -2.618E+00    0.000E+00   2.204E+02    0.000E+00   0.000E+00   0.000E+00
206 -3.398E+02    5.687E-05   5.955E+03    0.000E+00   0.000E+00   0.000E+00
```

6.7.7 [WRITE] Cutdown D3PLOT/PTF file



This figure (right) shows the menu for writing a compressed set of PTF files.

This option can be used to generate a new set of PTF/D3PLOT files for a model that contain only a subset of the Parts and States in the original model.

As well as reducing the size of the new PTF/D3PLOT files by reducing the number of PARTs and STATEs it is also possible to remove some of the data components from the files.

This operation can also be performed in batch using the dialogue commands:

`/UTILITES PTF_COMPRESS`

See Appendix VII for the full set of commands.

SELECT_PARTS This option can be used to select a subset of the model. After selecting the PARTs that are going to be written to the new set of PTF files the **PTF State Size** will be recalculated to show the size of each PTF state in the new files.

SELECT_STATES This option can be used to select a subset of the STATEs in the original model for output to the new set of PTF files.

As well as selecting a subset of PARTs and STATEs it is also possible to turn off the output of some data components to the new PTF files. Depending on what data componets and entity types the original model contains the table in the bottom section of the menu can be used to either select or deselect optional data components for output.

Due to the way the PTF file format works it is not always possibel to turn a data component on/off for a single entity type. If for example your model contains both Shells and Thick Shells then both the Stress Tensor and Plastic Strain must be either be truned on or off for both types of elements. Similarly the Strain Tensor must apply to all Solids, Shells and Thick Shells.

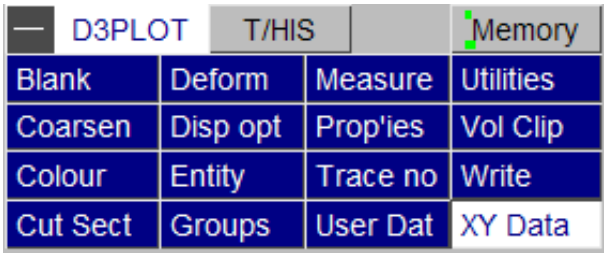
As data componets are turned on/off the **PTF State Size** will be recalculated to show the size of each PTF state in the new files.

INCLUDE NODES FOR ZTF ITEMS The PTF/D3PLOT files only contain information for the basic LS-DYNA element types. If you also have a XTF or XTF file then D3PLOT will use the extra information in these files to draw Springs. Joints etc.

If you use select a subset of PARTs for output it is possible to select a PART that is attached to one end of a Spring without selecting the PART connected to the other end of the spring. If this option is selected then D3PLOT will automatically ensure that the nodes at the end of any springs (or any other elements) that are no-longer attached to PARTs are also output to the new PTF files so the items cna be drawn correctly.

Next section

6.8 XY_DATA Drawing numerical data as XY plots and/or writing it to file



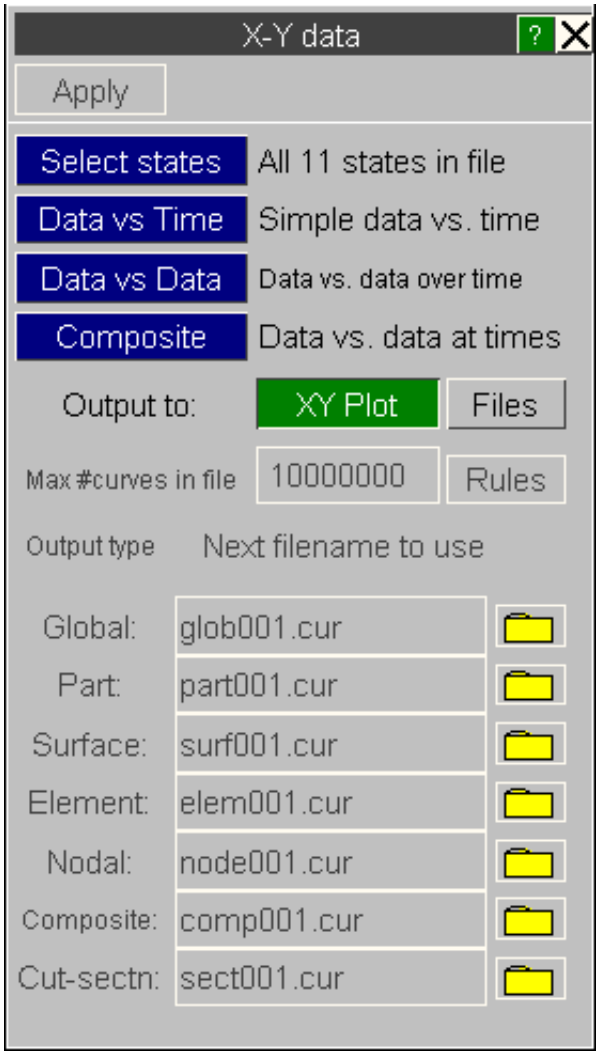
The **XY_DATA** command lets you extract scalar data values in a way similar to **WRITE**, but in this case over a range of times, and to plot them and/or write them to file.

XY_DATA is similar in principle to **TH_DATA**, the T/HIS link, but it operates on low-frequency data extracted from the plot database files (.ptf etc) rather than high frequency data from the time-history ones (.xtf etc). The **TH_DATA** command is described in section 6.12.

Extracted values are buffered in backing store, and remain there available for subsequent replotting until superseded by further data extraction. This backing store is limited in size, but it should be adequate to cope with extracting data for at least 500,000 items.

Data can only be extracted for one model at a time. If your run has more than one model currently open you will be forced to choose the model upon which to operate using the model "tabs".

6.8.0 Using the XY_DATA commands.



The **XY_DATA** main menu is shown right.

Select States Defines the time states used for output. (By default all states are preselected)

Three different data extraction and plotting methods are provided, and are descrived in more detail below:

(1) **Data vs Time** Plots data (Y axis) vs time (X axis)

(2) **Data vs Data** Plots data (X and Y axes) over a range of times

(3) **Composite** Plots data (X and Y axes) for a range of items at a time.

Output may be sent to either or both of:

XY Plot On screen graphical tool that displays curves. It permits curve selection, zooming, labelling and symbol display; it also allows you to select a subset of curves to be written to file.

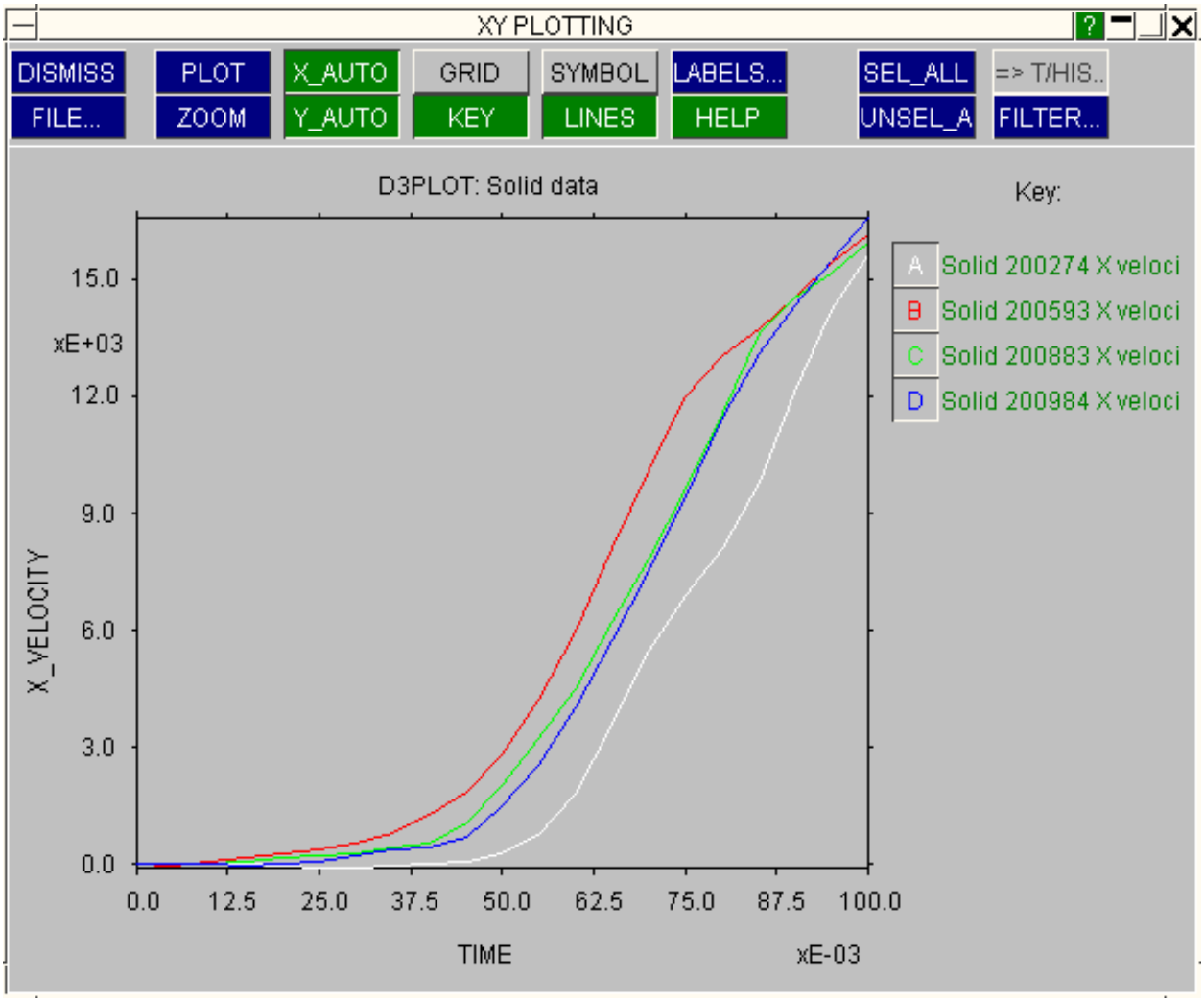
Files ASCII format files written in Oasys "curve file" (.cur) format.

The default filenames will become "live" if you select this option, and you may use these or define your own names.

The data requested are extracted from the complete states as they are read in, and stored on scratch backing store. When everything has been read in then the file(s) are generated and the XY plot drawn according to what output has been requested. XY plot management is described in Section 6.8.4, and file management in Section 6.8.5.

The data on backing store is "remembered" until you leave the programme, or overwrite it by reading in some more results. So once a set of data has been created you can exit and re-enter **XY_PLOT** at will to review it.

Typical graphical output from **XY_PLOT** is shown in the figure below



A typical file (the results for Material 1 above) is shown below. This is written in Oasys T/HIS "Curve" file format (see Section 4.8.5.5 for a description of this format):

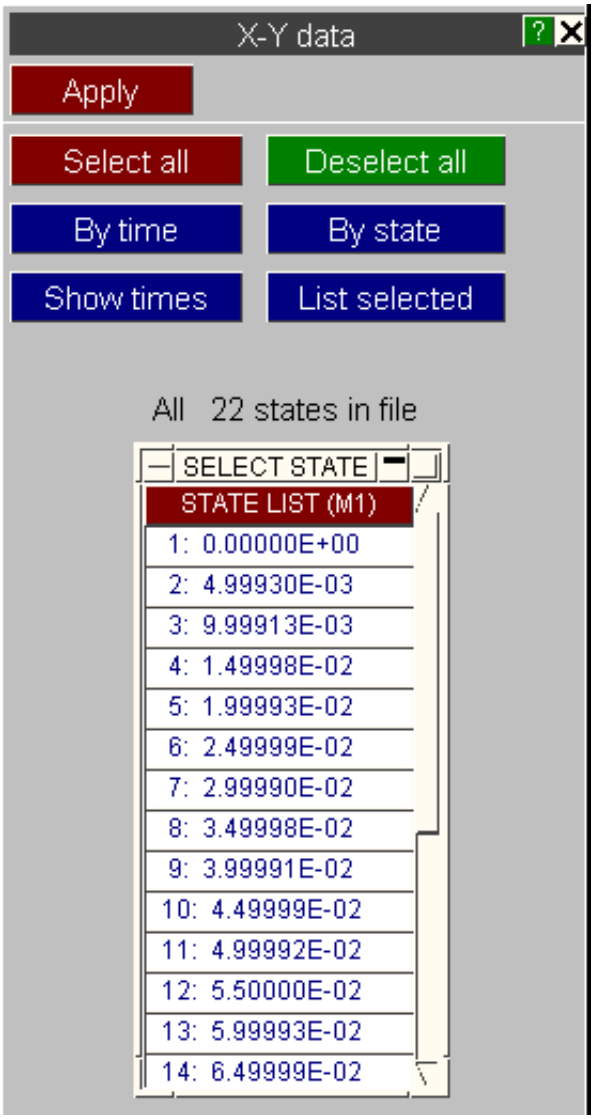
```
$ ---------------------------------------------------------
$ D3PLOT "T/HIS_INTERFACE" output file. (22-Nov-95 20:22:11)
$ Database file: /users/dyna70/test/lg09
$ Title: lg09 : Large Test 9: Belted sled test
$ ---------------------------------------------------------
```

D3PLOT: Material data
TIME
KE_KINETIC_ENERGY

Material 1	Kinetic energy
.000000E+00	7.222228E-05
2.490001E-03	1.272031E+01
4.990020E-03	3.979501E+01
7.490039E-03	6.687062E+01
9.999956E-03	9.656306E+01
1.249986E-02	1.276856E+02

1.499976E-02	**1.723507E+02**
1.749984E-02	**2.691978E+02**
1.999997E-02	**4.552962E+02**
2.249011E-02	7.637439E+02
2.499991E-02	1.173867E+03

6.8.1 SELECT_STATES
Choosing the complete states to be used for output.



The figure (right) shows the **SELECT_STATES** control panel.

By default all states in the file are selected for **XY_PLOT** output, but you may select any sub-set that you wish.

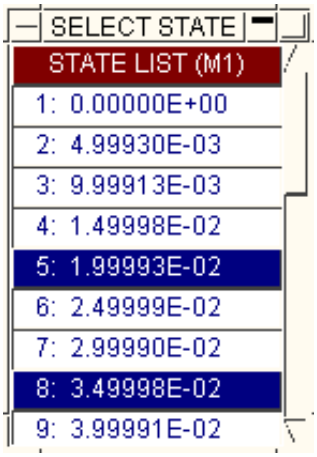
Once your selection is complete use **APPLY** to return to the main **XY_PLOT** menu.

The commands to select and modify the states selected for **XY_PLOT** output are identical to those used for animation. (Ref Section 4.4.2.) However the internal storage of selected state numbers is totally separate: states selected for XY plotting will not affect those stored for animation, and vice-versa.

States may be selected using **BY_TIME** (start rime, interval, end time) or **BY_STATE** (start state, interval, end state)

It is also possible to select and deselect times from the **STATE_LIST** menu. Click on an undepressed state to select it, or on a depressed state to deselect it.

This menu always the shows what has been selected, regardless of the selection method used. And you can mix selection methods: for example you could select states by time, then modify the result in this menu.

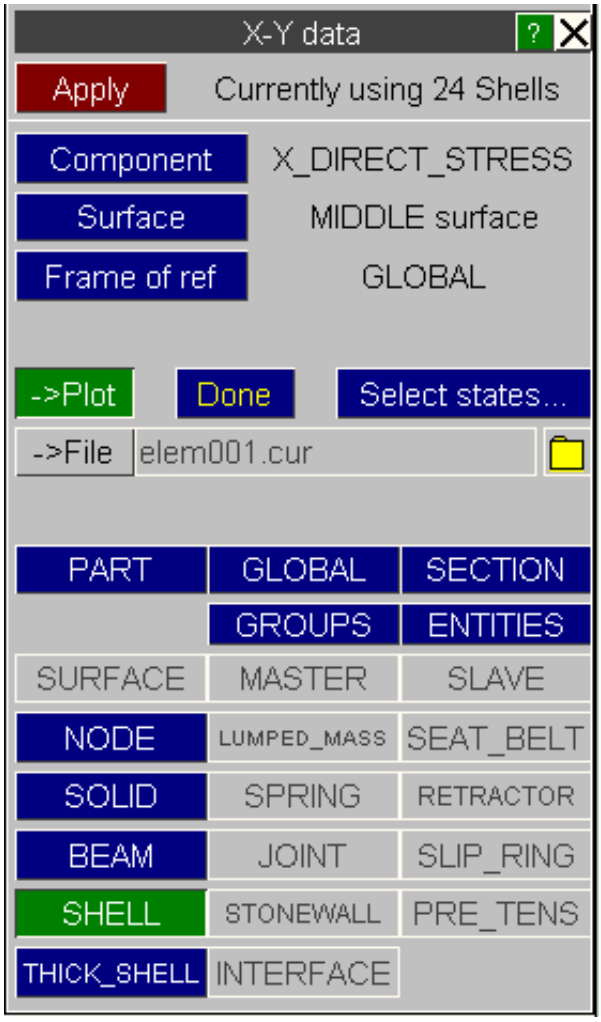


Important notes on internal state lists:

Order is important: The order in which states are selected is significant. If you pick entries #1, #4, #2 from the menu here, or when defining a <list>, then that is the order in which they will be processed. Implicit in this is the fact that states can be repeated, for example the sequence <#1, #2, #3, #2, #1> is quite legal, subject to the space restrictions below.

Space is limited: The amount of space set aside for storing state lists is 2x the number of states available. Thus you could store the sequence <#1 - #n - #1>, but no more than this.

6.8.2 **Data vs Time** Generating data vs time.



This is the simplest form of XY plotting:

- You select a list of items (here shells have been chosen)
- You select a data component (here X Direct Stress)
- Optionally select shell surface and frame of reference

Apply will then extract and plot the data.

For each item chosen, here shells, a curve will be generated of

X axis: Time
Y axis: Selected data component

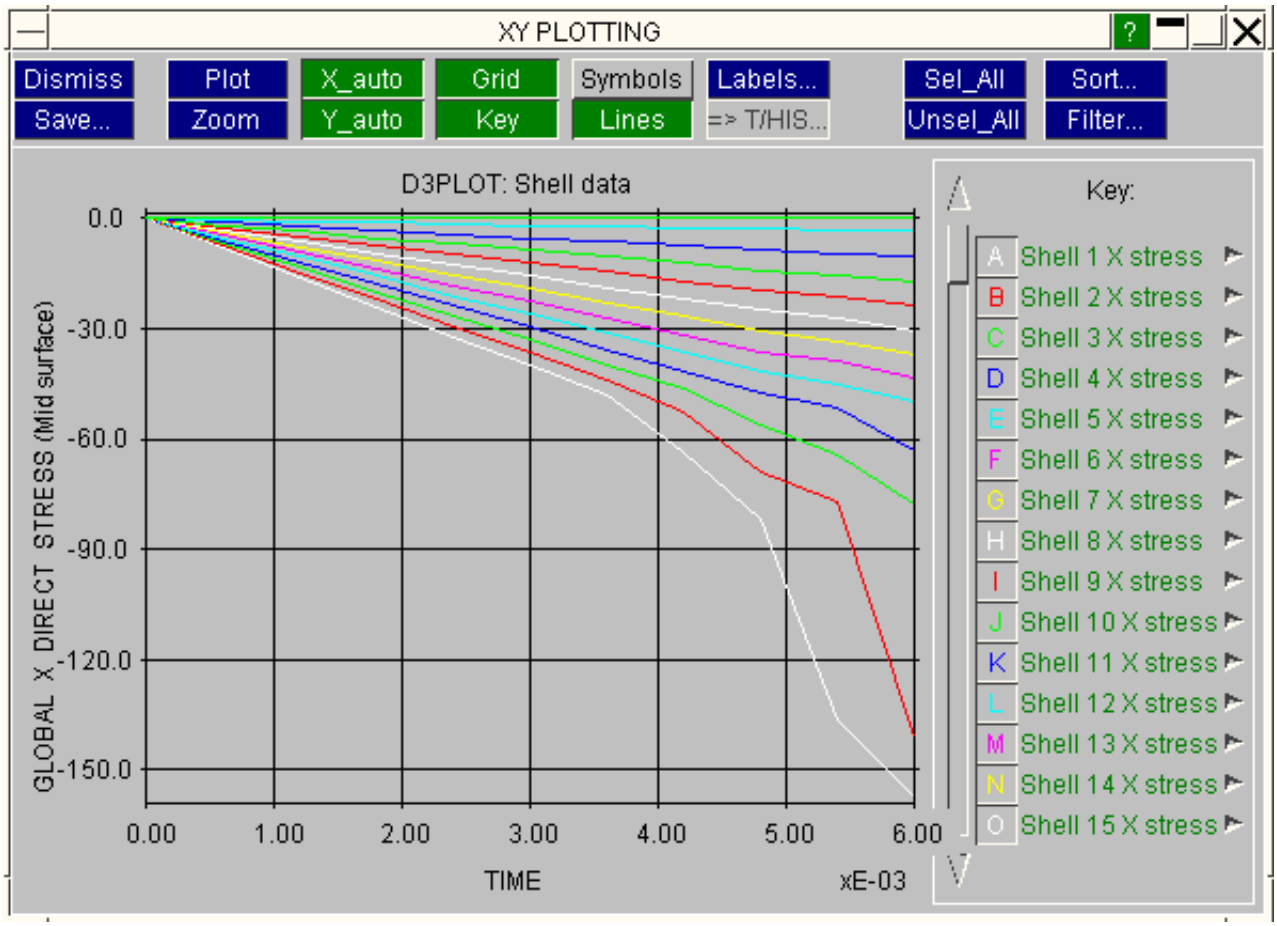
-> Plot Toggles display to the graphical XY plotting tool

-> File Toggles output to a curve file, and you can define the filename.

Select states Allows you to define different states for output

Done Returns to the main XY plotting menu

Here is the output generated by the panel above:


XY PLOTTING
Dismiss
Save...
Plot
Zoom
X_auto
Y_auto
Grid
Key
Symbols
Lines
Labels...
=> T/HIS...
Sel_All
Unsel_All
Sort...
Filter...
D3PLOT: Shell data
GLOBAL X DIRECT STRESS (Mid surface)
0.0
-30.0
-60.0
-90.0
-120.0
-150.0
0.00
1.00
2.00
3.00
4.00
5.00
6.00
TIME
xE-03
Key:
A Shell 1 X stress
B Shell 2 X stress
C Shell 3 X stress
D Shell 4 X stress
E Shell 5 X stress
F Shell 6 X stress
G Shell 7 X stress
H Shell 8 X stress
I Shell 9 X stress
J Shell 10 X stress
K Shell 11 X stress
L Shell 12 X stress
M Shell 13 X stress
N Shell 14 X stress
O Shell 15 X stress

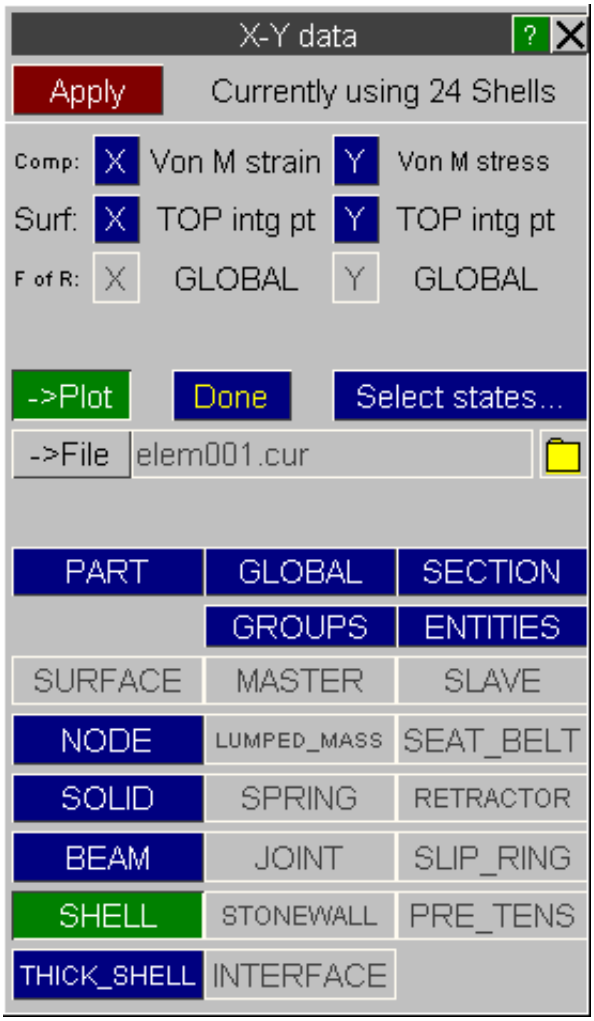
6.8.3 **Data vs Data**: Both X and Y data components against time.

This is very similar to the Data vs Time panel above, except that you choose data components for both X and Y axes.

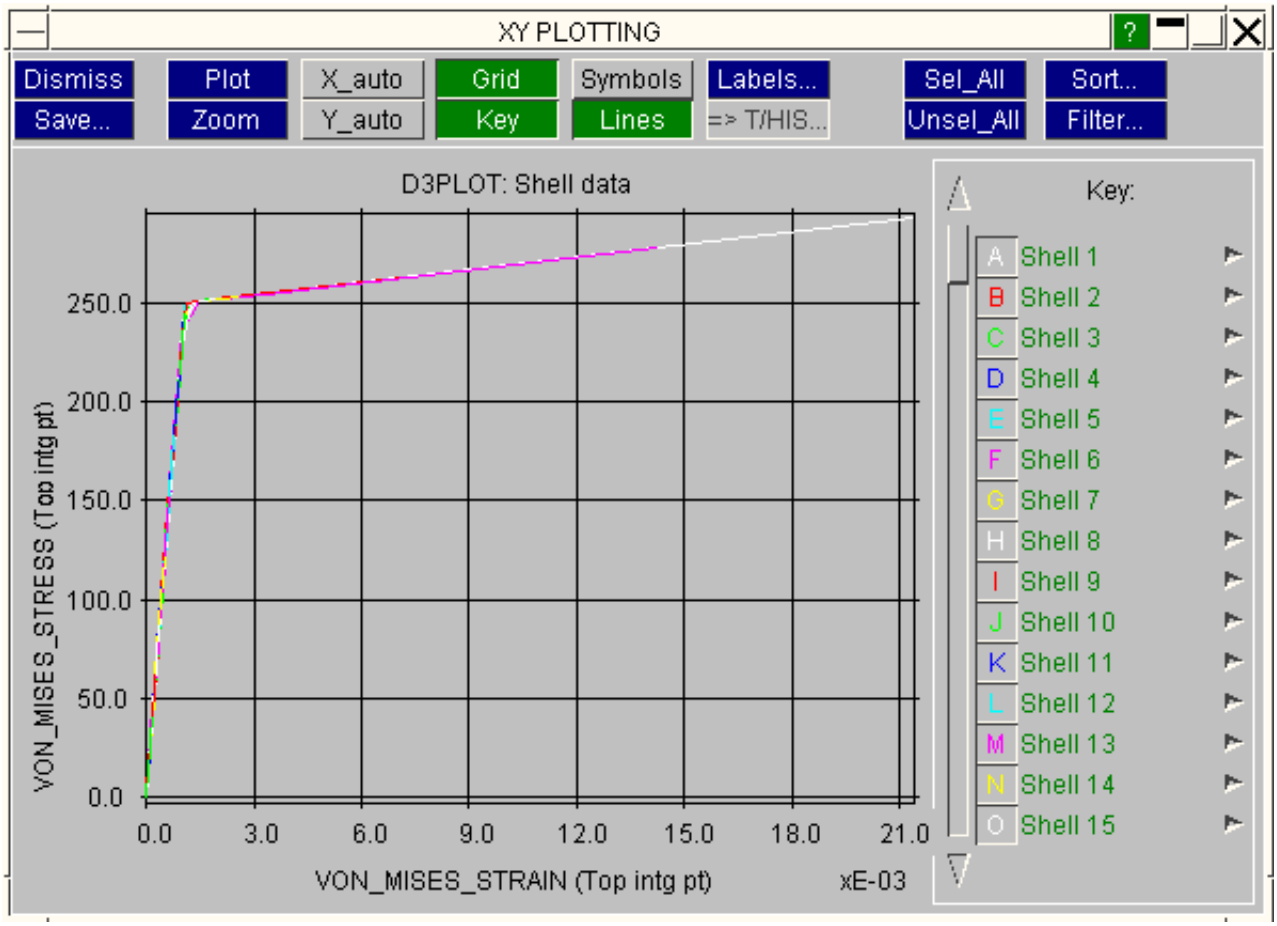
Each item (here shells) still forms a curve, but the data points are <X data> vs <Y data> at each selected state.

For example you might wish to plot the stress vs strain history for a set of elements, in which case you might choose:

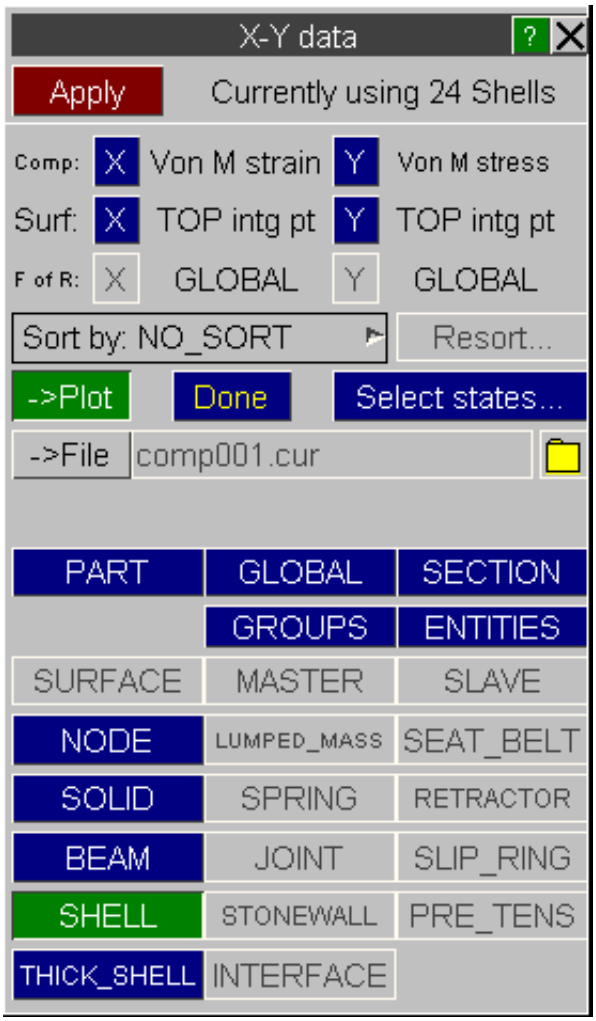
- X Axis: Strain
- Y Axis: Von Mises stress



Here is an example of output from such a plot, showing Von Mises Strain (X axis) against Von Mises Stress (Yaxis). This shows the classic stress vs strain curve for mild steel.


XY PLOTTING
Dismiss
Save...
Plot
Zoom
X_auto
Y_auto
Grid
Key
Symbols
Lines
Labels...
=> T/HIS...
Sel_All
Unsel_All
Sort...
Filter...
D3PLOT: Shell data
VON_MISES_STRESS (Top intg pt)
250.0
200.0
150.0
100.0
50.0
0.0
0.0
3.0
6.0
9.0
12.0
15.0
18.0
21.0
VON_MISES_STRAIN (Top intg pt)
xE-03
Key:
A Shell 1
B Shell 2
C Shell 3
D Shell 4
E Shell 5
F Shell 6
G Shell 7
H Shell 8
I Shell 9
J Shell 10
K Shell 11
L Shell 12
M Shell 13
N Shell 14
O Shell 15

6.8.4 **COMPOSITE_DATA** Data vs data for a list of items over a range of times.



"Composite" data is similar to "Data vs Data" except for the vital difference that:

- Each curves contains (x,y) data for the list of items selected, in order, at a particular time.
- Therefore the number of curves = the number of states selected, not the number of items
- And the number of points in each curve = the number of items selected.

This plotting mode is useful for displaying the variation of a quantity across space in a model over a range of times, as the examples below show.

IMPORTANT: The order in which points are written in composite curve is the order in which they are defined, unless subsequently sorted.

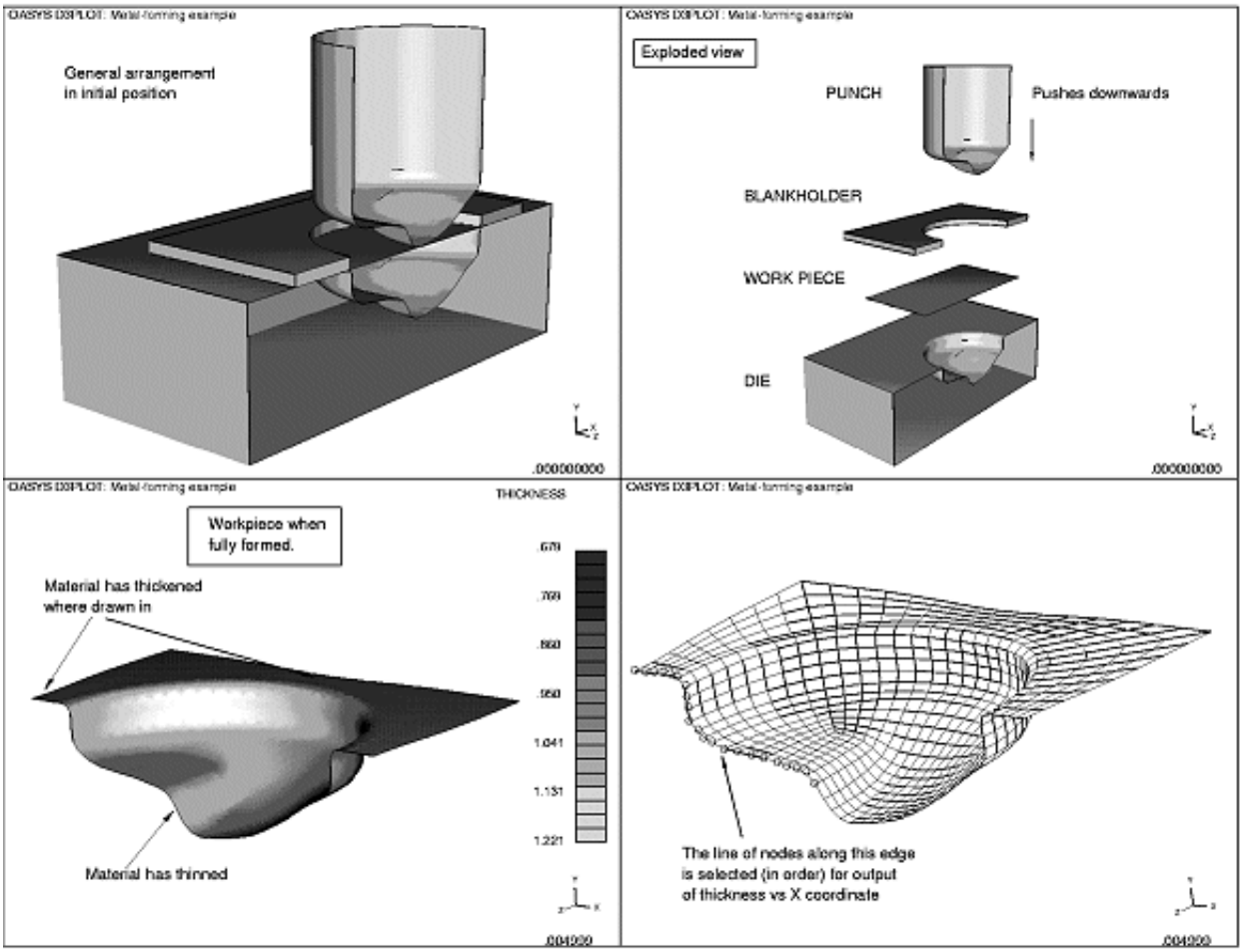
- If you screen-pick items they will appear in the order picked up to a limit of 1000 items
- If you select by area, polygon on "all" they will appear in ascending label order

Sort by: sorting data points.

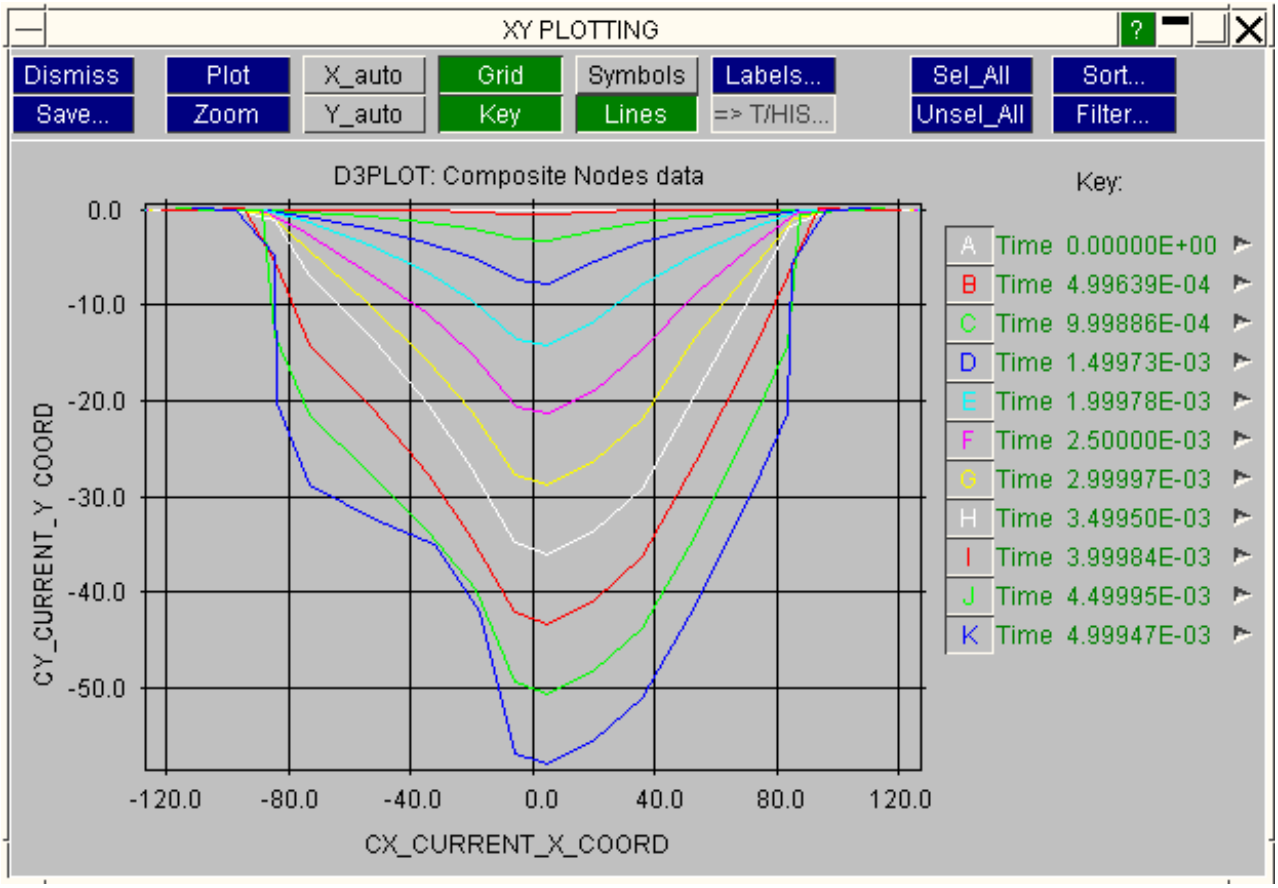
When points are picked by screen area, or some other unordered means, their order is usually "wrong". You can use the **Sort by** options to correct this, and these are described below.

6.8.4.1 Example: Displaying profile change during metal-forming.

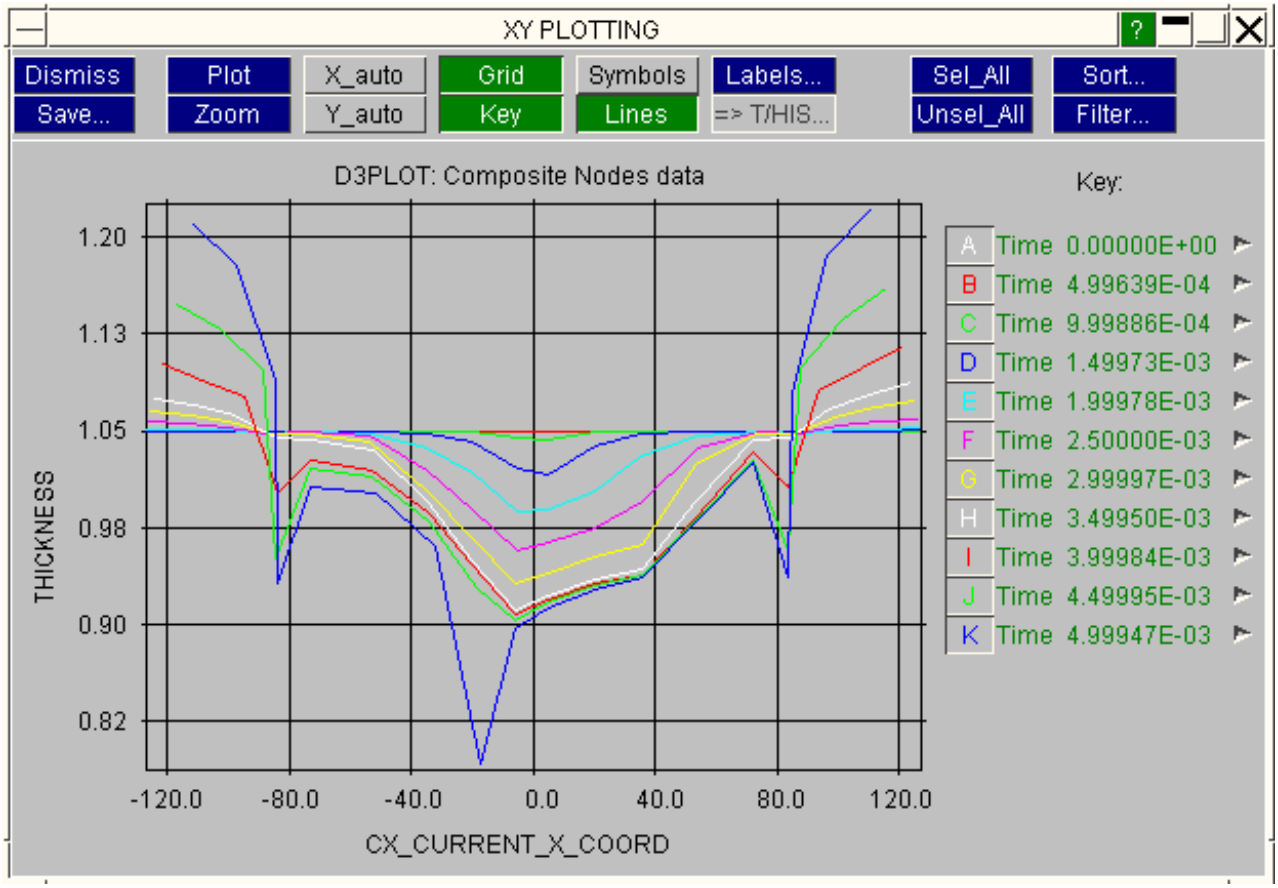
The figures below show a metal-forming operation, and the way composite plots could be used to interpret the results. Clearly the order in which the list of nodes (along the edge of the workpiece) is defined is important for the composite plots below.


General arrangement
in initial position
Exploded view
PUNCH
Pushes downwards
BLANKHOLDER
WORK PIECE
DIE
Workpiece when
fully formed.
Material has thickened
where drawn in
Material has thinned
THICKNESS
The line of nodes along this edge
is selected (in order) for output
of thickness vs X coordinate

Composite plot of X vs Y coordinates showing formed profile



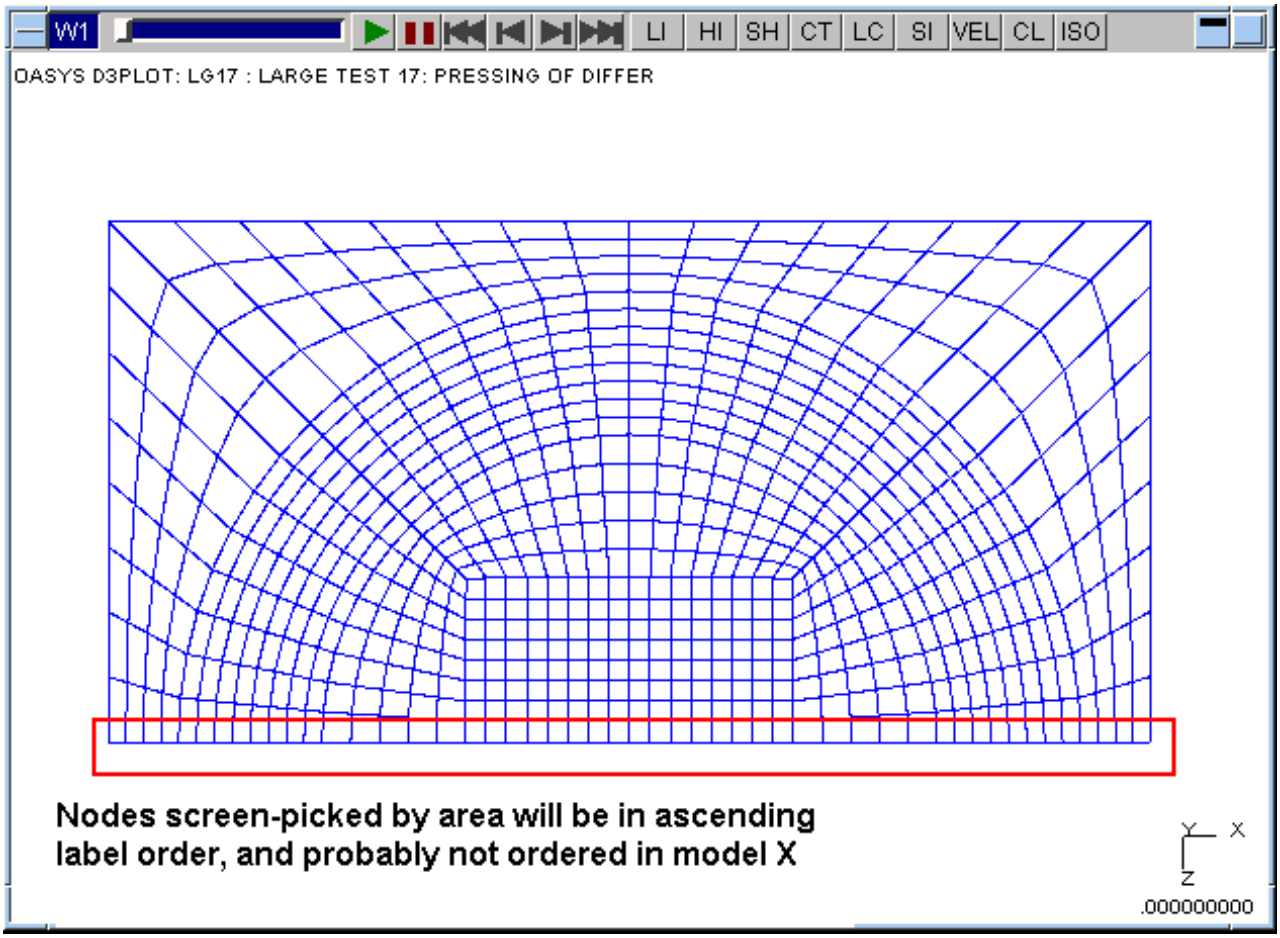
Here is a composite plot of thickness vs X coordinate for the same model



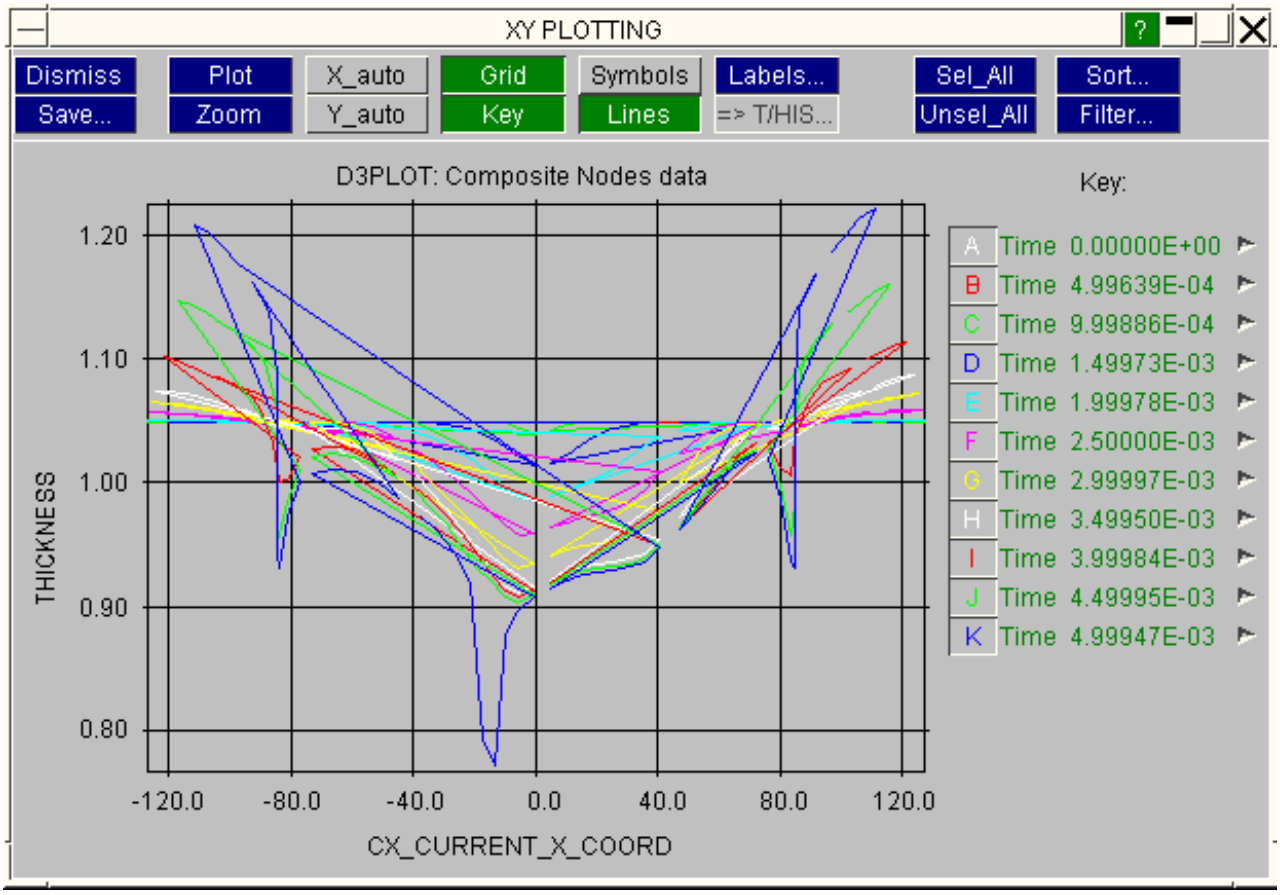
6.8.4.2 **Sort By**: Sorting data points in Composite plots

In the examples above nodes along the workpiece edge were selected carefully, and in order, by screen picking. This can be time-consuming so an alternative is to select by area and then to sort by some criterion. The example below shows what may happen, and how to correct it.

Screen-picking nodes by area



Sure enough, some of the nodes are not ordered in ascending X coordinate, giving a muddled plot.

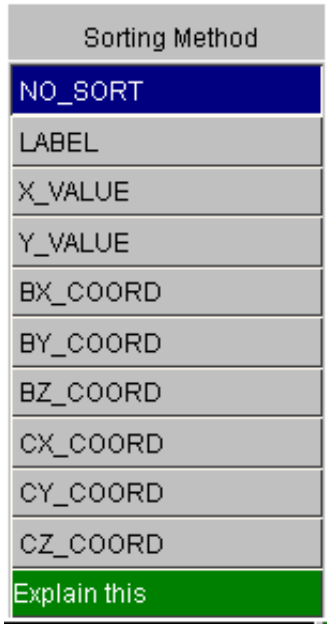


By default no sorting takes place, but you can sort by any of the options in this menu.

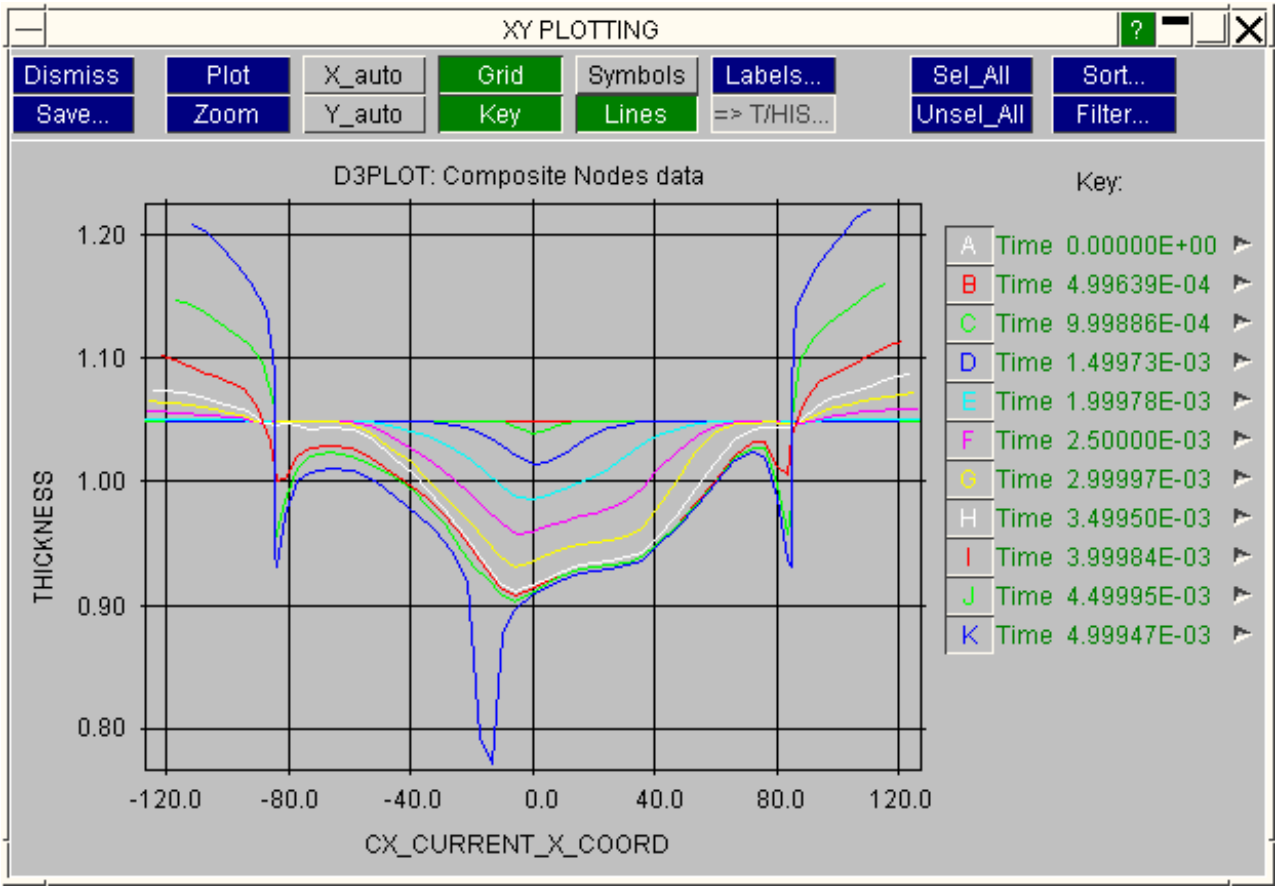
In this case sorting by **CX_COORD**, the current X coordinate, will work for the plot above, and the plot below shows how this corrects the image.

Note that:

- The current **Sort by** option remains current until changed. Any data subsequently extracted will be sorted in this way.
- You can resort data currently cached in backing store at any time



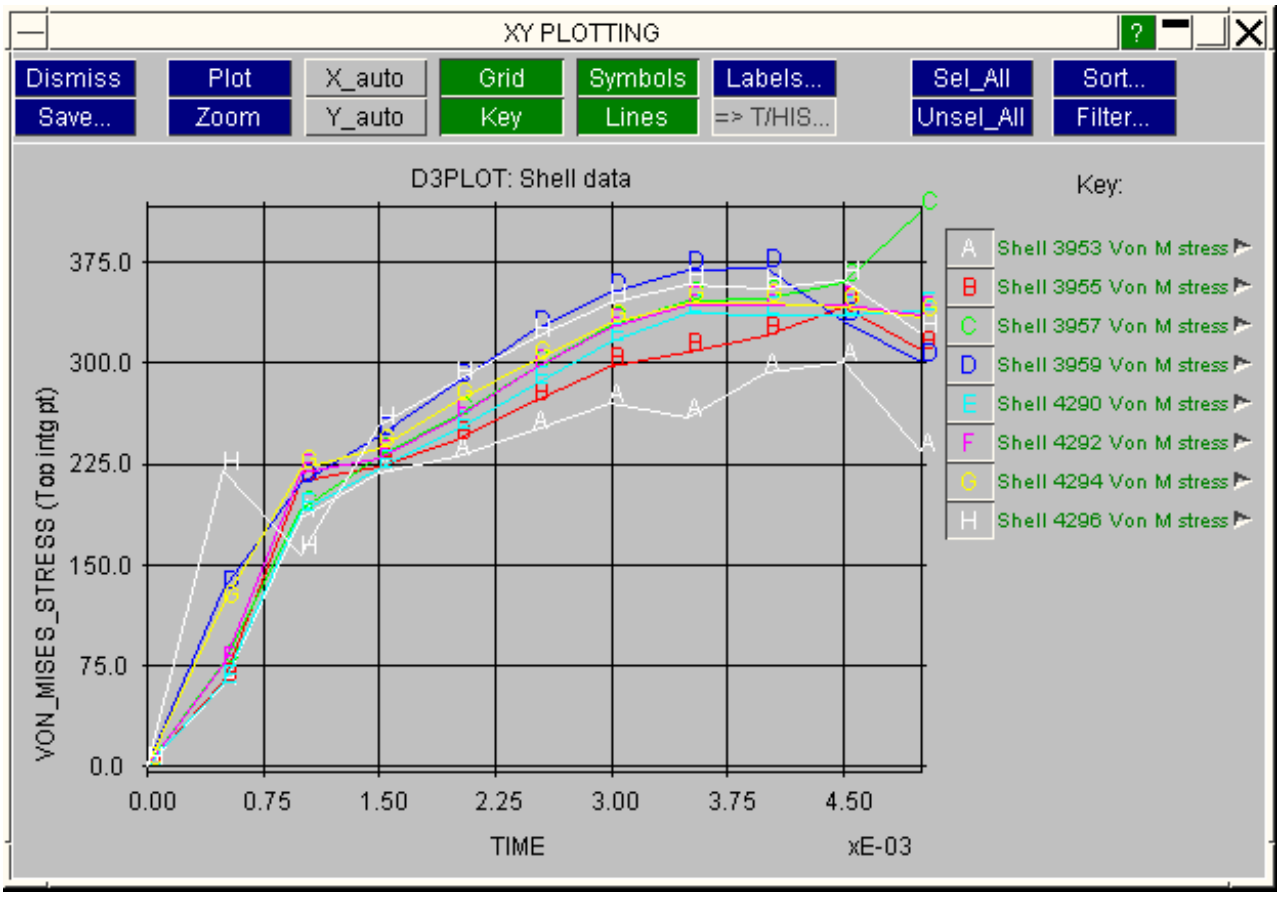
After sorting by Current X coordinate the points are back in a sensible order.



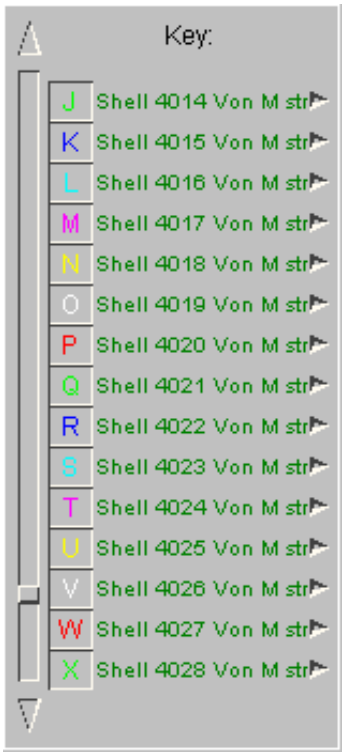
Next section

6.8.5 Using the XY graphical plotting tool.

When XY plots are requested they are drawn using a standard XY plotting tool. This operates using a menu system window, so it is not available if the screen menu interface is not being used. The figure below shows a typical XY data plot.



6.8.5.0 Selecting the curves to be plotted


Key:
J Shell 4014 Von M str
K Shell 4015 Von M str
L Shell 4016 Von M str
M Shell 4017 Von M str
N Shell 4018 Von M str
O Shell 4019 Von M str
P Shell 4020 Von M str
Q Shell 4021 Von M str
R Shell 4022 Von M str
S Shell 4023 Von M str
T Shell 4024 Von M str
U Shell 4025 Von M str
V Shell 4026 Von M str
W Shell 4027 Von M str
X Shell 4028 Von M str

By default all curves are plotted, but you can select and deselect individual curves by clicking on their label buttons (under **Key:**). If there are too many curves to fit into the vertical space availble a scroll-bar is added as shown to the right.

Only those curves selected for plotting are drawn when the next **PLOT** command is issued. You can use **SEL_ALL** and **UNSEL_ALL** to (de)select all of them, and the **FILTER** options to select and deselect curves by colour and/or label: see below.

The colours (white, **red**, **green**, blue, **cyan**, **magenta**, **yellow**) and letters (A-Z) of curves are fixed and cannot currently be changed. The letters are used as "symbols" when symbol display is sturned on, as shown in the plot above.

Obtaining information about curves.

Key:
J Shell 4014 Von M str
K Shell 4015 Von M str
L Shell 4016 Von M str
M Shell 4017 Von M str
N Shell 4018 Von M str
O Shell 4019 Von M str
P Shell 4020 Von M str
Shell 4015 Von M stress
X max/min: 0.49995E-02 0.0000
Y max/min: 362.60 0.0000

You can use the popup menus against any "live" curve in the scrolling key to obtain the max/min X and Y axis data for that curve.

Here curve **K** has been selected.

6.8.5.1 Top commands

Dismiss Closes the XY plotting window. Data is left unchanged on backing store.

Save... Writes the currently selected curves only to "curve" files. The syntax and other conventions for these files are described in Section 6.8.6.

Plot Redraws the current graph with only those curves selected for plotting.

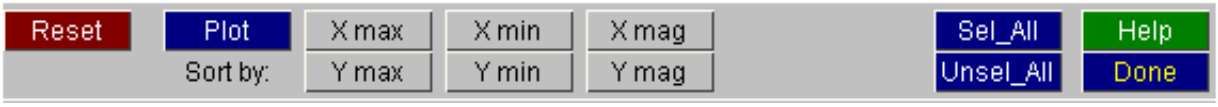
Zoom Uses the cursor to zoom in on a rectangular area of the graph, and enlarges this to fill the whole window. Both X and Y scales become set explicitly, ie taken out of automatic mode, by this command.

X_auto Resets the X scaling to automatic.

Y_auto Resets the Y scaling to automatic.

Grid	Toggles the grid at tick mark intervals on/off. (Default ***off***)
Key	Toggles the curve "Key" listing on/off. (Default on)
Symbols	Toggles symbol (ie letter) display at points on/off. (Default ***off***)
Lines	Toggles lines between points on/off. (Default ***on***)
Labels	Lets you redefine title and axis labels.
=> T/HIS	Copies selected curves to linked T/HIS (when running) (See below)
Sel_All	Selects all curves for plotting.
Unsel_All	Deselects all curves for plotting.
Sort...	Sorts the order of appearance of curves in the key
Filter...	Filters visible curves by colour and symbol (See below)

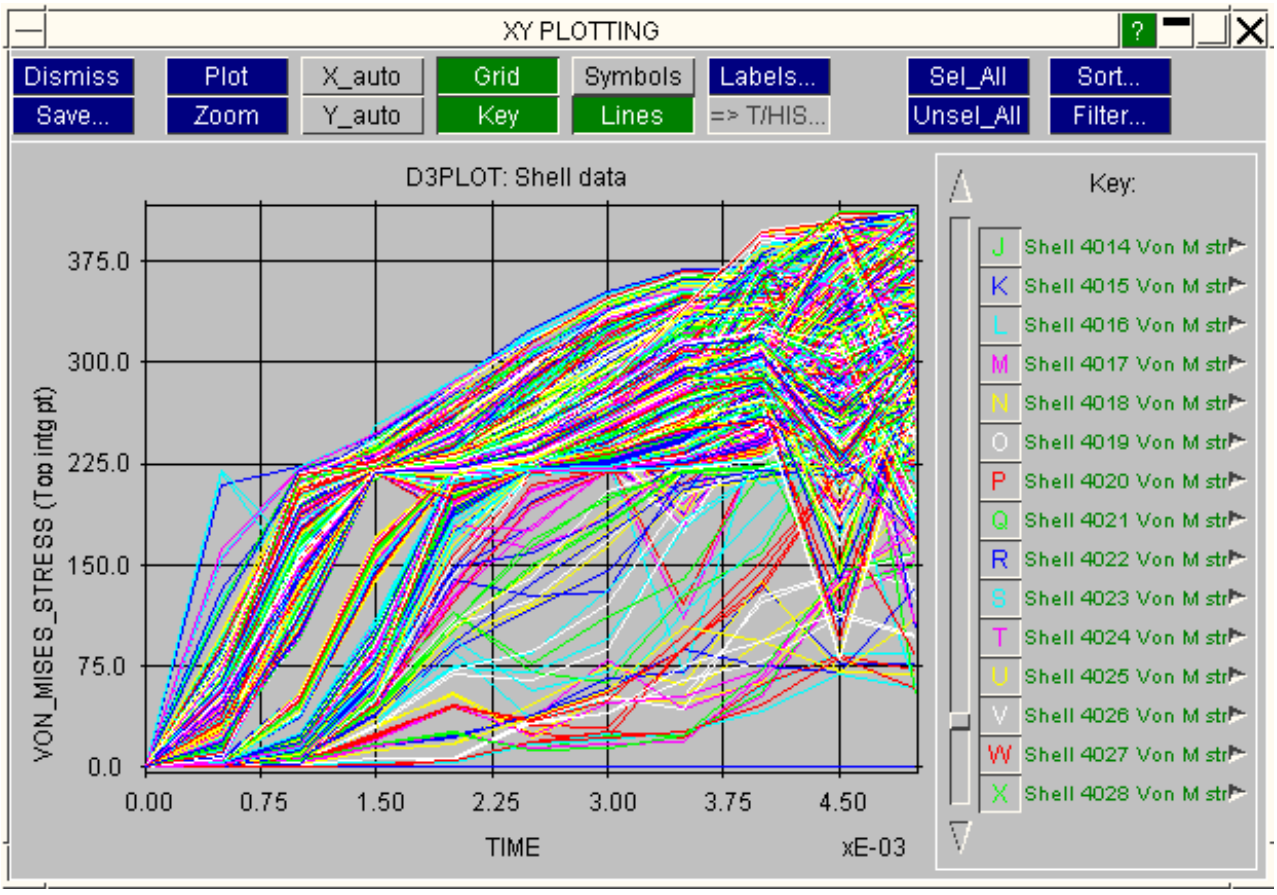
6.8.5.2 **Sort...** Sorting the order of appearance of curves in the key



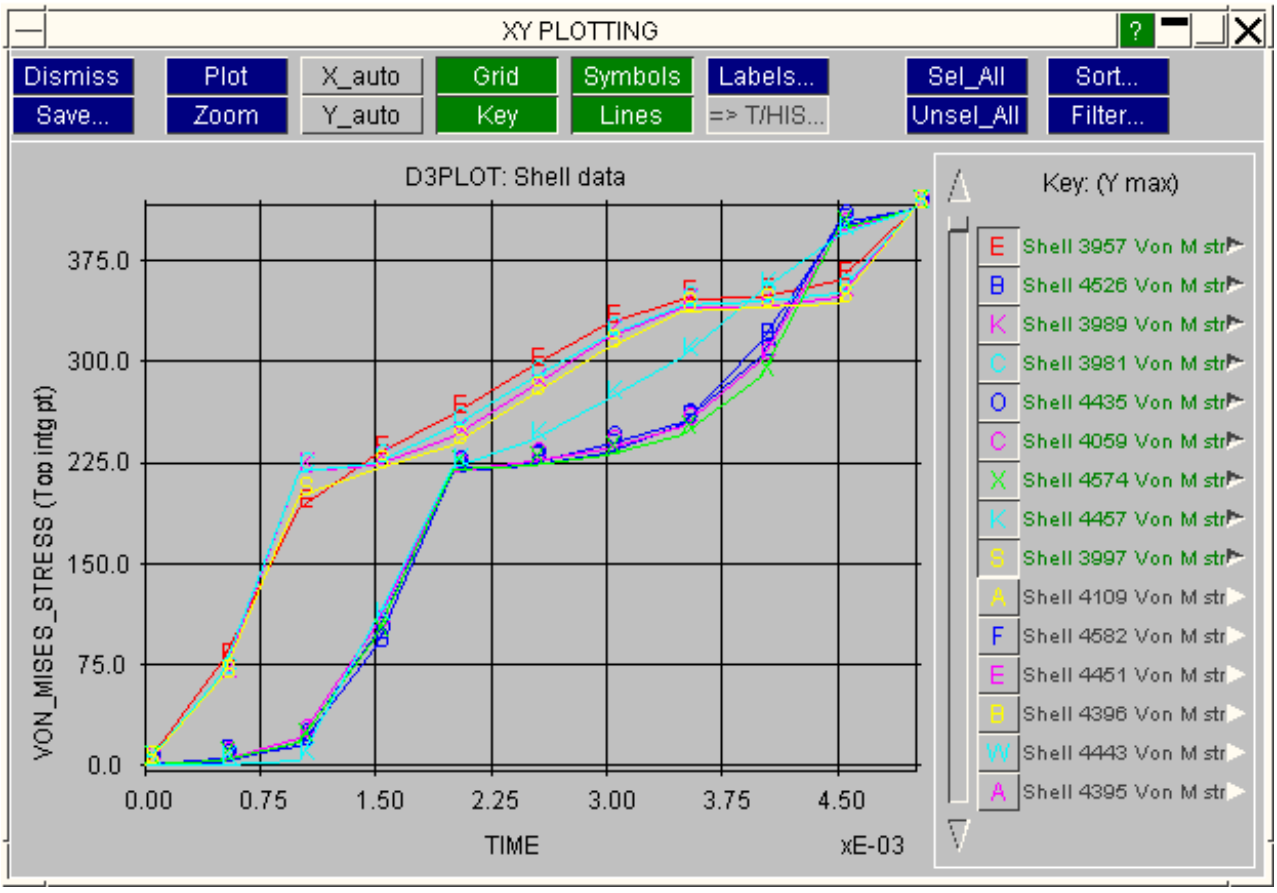
By default the curve "key" will appear in order of item definition, sorted by ascending label number. However when confronted by a plot with many curves often it is useful to be able to pick out the largest or smallest, and this can be achieved with **Sort....**

You can sort by any of the criteria X max/min/mag, Y max/min/mag, and the effect will be to reorder the curve key as appropriate.

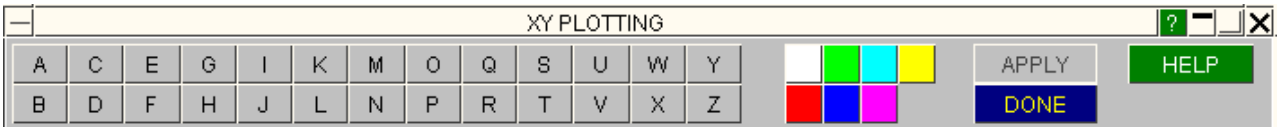
For example: here is a plot of von Mises Stress for 4624 shells over time - confusion - which is the highest value?



Using **Sort...** by Y max reorders the key with the top Y values at its top, making it possible to turn on just the top few curves, which are plotted below.



6.8.5.3 **FILTER** Commands



When you have many curves the **FILTER** option is another way of limiting what is displayed.

Clicking on a letter selects only the curves with that symbol, and on a colour only those of that colour. You can use both at once to restrict selection to just curves of that letter/colour combination. Use **APPLY** to plot just those selected, and **DONE** to return to the main plotting options menu.

6.8.5.4 **=> T/HIS** command: Export curves to linked T/HIS session

If the D3PLOT <=> T/HIS link has been invoked it is possible to export selected curves to the linked T/HIS session.

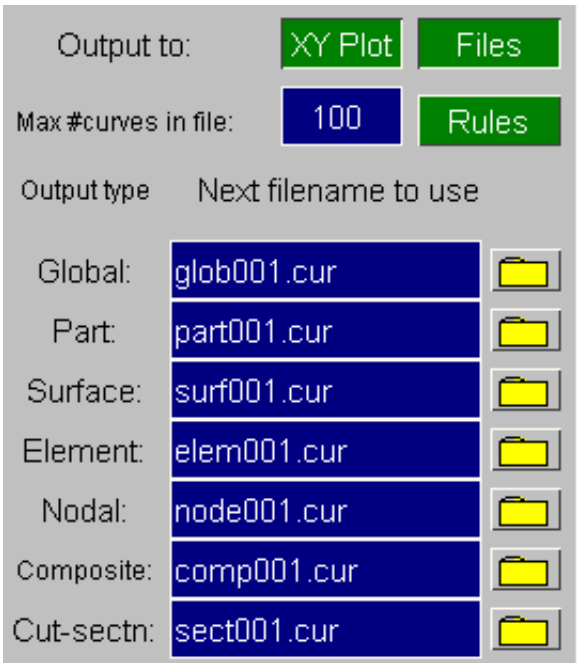
The number of free curves available in T/HIS is shown, and you can choose which of the local XY_DATA curves to export. These will become the next free #n curves in T/HIS.

(If linked T/HIS has not been invoked via the THDATA command the **=> T/HIS** button will be greyed out, and this function will be unavailable.)

6.8.6 Managing "curve" file output.

By default output from **XY_DATA** is written to "curve" file as well as plotted.

This is controlled by the **FILES** command on the main control panel: when turned off output to file is suppressed. (Nb:You can also write files from the XY plotting window using the **FILE...** command there.)



6.8.6.1 File naming conventions

All "curve" files **must** have a name in the form **<name>NNN.<extension>**, where:

<name> is any sensible character string, but see conventions below;

NNN is a 3 or 4 digit number using leading zeros if required;

<extension> is any sensible character string. ".cur" is recommended.

The reason for this syntax is that more than one file may be generated by an output command, and some sort of systematic naming convention is required.

By default the following filenames are used:

For global model derived data vs. time: **glob001.cur**

For part derived data vs. time: **part001.cur**

For contact surface derived data vs. time: **surf001.cur**

For element derived data vs. time: **elem001.cur**

For nodally derived data vs. time: **node001.cur**

For any **COMPOSITE** data: **comp001.cur**

You can change them if you wish, but it is recommended that you do not. If you omit the **NNN** part of the filename string D3PLOT will insert it for you.

6.8.6.2 Maximum number of files in a sequence.

Implicit in the naming convention above is the sequence

<name>001.cur to **<name>999.cur** followed by

<name>1000.cur to **<name>9999.cur**

Thus there is a maximum of 9999 files in a sequence.

Each new file output operation starts a new curve file, and D3PLOT will enforce the addition of integers to the base filename to ensure that each file has a unique name. One consequence of this is that it is not possible to overwrite a curve file from within D3PLOT: if you want to reuse a particular filename you will need to delete it externally to the programme.

6.8.6.3 Defining the maximum number of curves in a file.

From V92 onwards the default is for 10,000,000 curves to be written to a "curve" file. When the limit is reached the current file is closed, and the next one in the sequence is opened for output. In practice this means that all output from a single curve output command will be written to a single file.

You can change this value to any positive integer with the **Max #curves in file:** value.

6.8.6.4 Rules for file output.

The following rules for file output are adopted:

(1) Each output command **always** starts a new file.

(2) Existing files are **never** overwritten. Before a new file is opened a test is made to see if one of the same name already exists and, if it does, the **NNN** value is incremented until a vacant filename is found.

6.8.6.5 Format of an Oasys T/HIS "curve" file.

This is an ASCII file which is organised as follows:

Line 1: Graph title (CHARACTER string)
Line 2:
X axis label (CHARACTER string)
Line 3:
Y axis label (CHARACTER string)
Line 4:
Curve label (CHARACTER string)
Line 5:
<X value> <Y value>(REAL numbers)
Line 6:
<X value> <Y value>(REAL numbers)
:

Line n: <X value> <Y value>(REAL numbers)
And if a second or subsequent curves are to be defined in the file:
Line n+1: CONTINUE (CHARACTER constant)

Line n+2: Graph title (CHARACTER string)

Line n+3: X axis label (CHARACTER string)

Line n+4: Y axis label (CHARACTER string)

Line n+5: Curve label (CHARACTER string)

Line n+6: <X value> <Y value>(REAL numbers)

:
And so on for further curves.

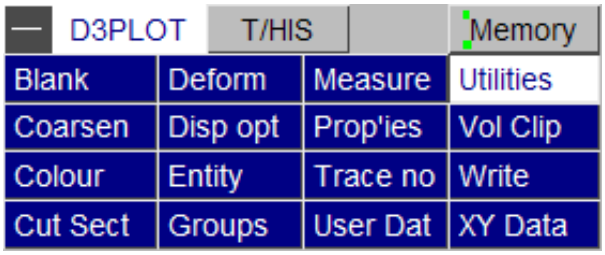
Further notes on Oasys T/HIS curve file format:

- Character strings should not exceed 48 characters.
- Numbers may be entered in free format, separated by spaces or commas. It is recommended that a field width of 20 be used (use a Fortran format statement something like 2e20.8) since this will then be compatible with Load Curve input in LS-DYNA.
- A curve may contain any number of <data> pairs.
- A file may contain any number of curves, separated by CONTINUE statements.

For more information see the Oasys T/HIS User's Manual.

Next section

6.9 **UTILITIES** Miscellaneous utility functions



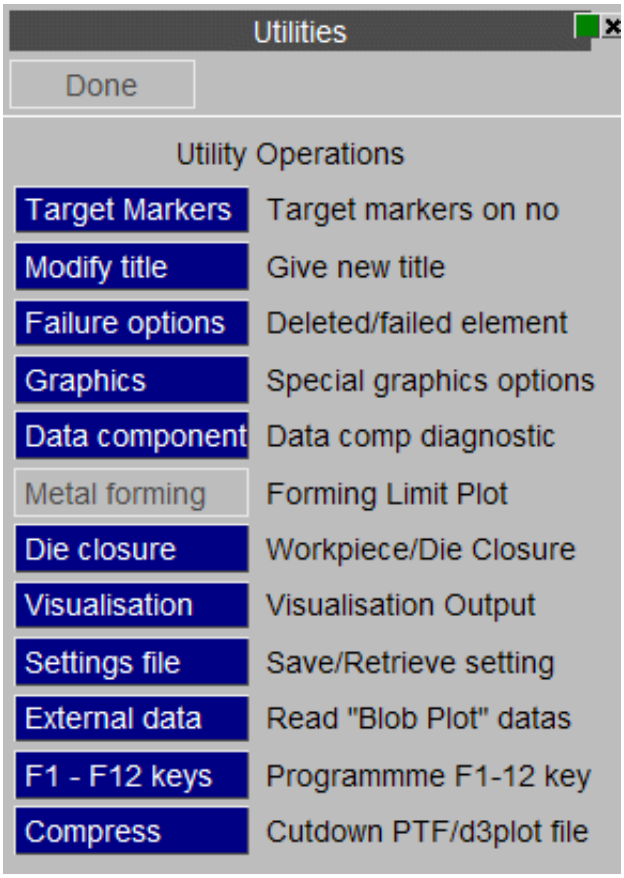
The **UTILITIES** menu is shown in the figure (right).

It provides a home for commands that have no other more logical place in the D3PLOT programme structure, and also for functions under development.

As the programme develops the latter commands will move out to their own menus.

Quick links to sections:

6.9.1 Target Markers
6.9.2 Modify Title
6.9.3 Failure Options
6.9.4 Graphics
6.9.5 Data Components
6.9.6 Metal Forming
6.9.7 Die Closure
6.9.8 Visualisation
6.9.9 Settings File
6.9.10 External Data
6.9.11 Function (F1 - F12) Keys
6.9.12 Compress



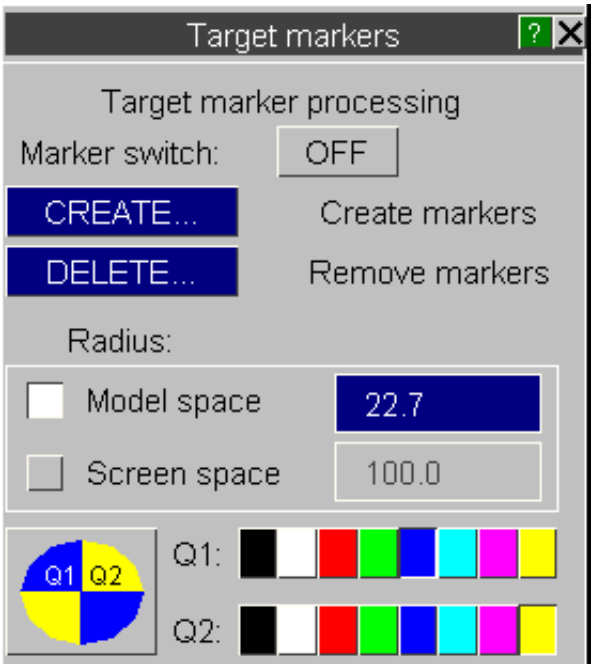
6.9.1 **TARGET_MARKERS** Adding “target” symbols on nodes.

The figure (right) shows the **TARGET_MARKERS** control panel.

Target markers are used to mimic the actual markers that are attached to objects when being tested. This allows direct visual comparison between points on high-speed film and analytical results.

Because markers are attached to nodes they can only be defined in one model at a time. If multiple models are present you will be forced to choose the model in which to define the markers.

Markers are stored as a "per model" attribute, so if you have multiple windows in a model the markers will appear in all of its windows.



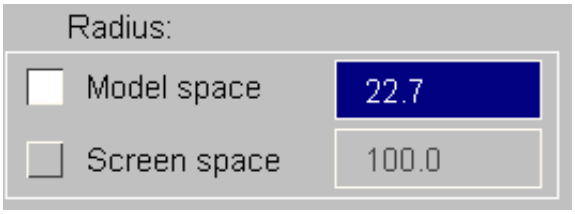
The figure (right) shows an example of a dummy model with target markers applied.

Markers are made up a circle with quadrants of alternating colours. On the greyscale reproduction used here they do not stand out very well, but in colour they are much more marked.

They are attached to nodes and are drawn only if their parent node is visible.

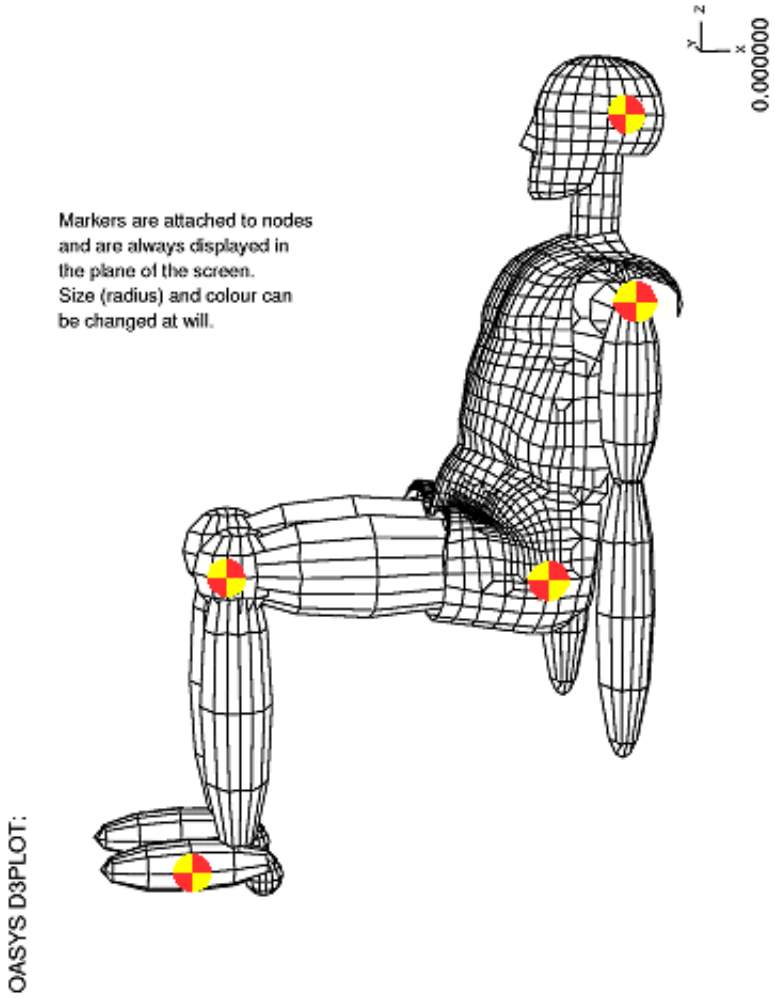
They are always drawn in the current plane of the screen regardless of the true orientation of any underlying mesh.

Marker Radius



If expressed in model space, the default, marker radius will be about 3% of the model overall dimension, and size will change as you zoom in and out.

If expressed in screen space marker radius will be a fixed size, independent of the model dimensions, and will not scale as you zoom in or out.


Markers are attached to nodes and are always displayed in the plane of the screen. Size (radius) and colour can be changed at will.
0.000000
OASYS D3PLOT:

6.9.1.1 **Marker switch:** Controlling display of target markers.

By default this switch is on, and any markers that are defined will be drawn when the plot is next updated. To suppress the display of markers without having to delete their definitions turn this switch off.

6.9.1.2 **CREATE** Creating target markers at nodes

This is very simple. Define a <list> of nodes using the standard range definition panels, and the next time the model is drawn (with the Marker switch on) target markers with the current attributes will be drawn on those nodes.

6.9.1.3 **DELETE** Removing target markers from nodes.

This is the exact opposite of **CREATE** above: define the <list> of nodes from which target markers are to be removed. It doesn't matter if a node is selected that does not currently have a marker attached.

6.9.1.4 **Radius** Defining the symbol radius

A default radius for markers (in model space units) is computed based on the model's longest dimension: you can alter this at will. This takes effect the next time you update the plot.

6.9.1.5 Setting quadrant colours

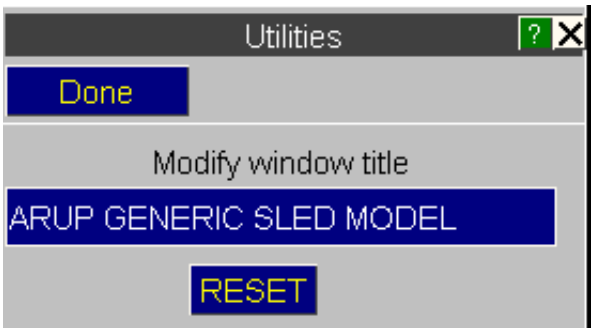
The default colours for the marker symbol are yellow and blue, but you can choose new colours for the two quadrants (**Q1** and **Q2**) from the standard palette given. These take effect the next time you update the plot.

6.9.1.6 Notes on target markers

Note 1: Target markers are always drawn in the plane of the screen, regardless of model orientation. On 3-D devices this results in the symbols being rotated as the image is rotated, and they can disappear when edge-on after a 90 degree rotation. To see them again you will have to issue an explicit redrawing command in the new orientation.

Note 2: Hidden surface removal is based on the coordinates of the parent node only: markers are drawn in full if the node is visible, otherwise not at all.

6.9.2 **MODIFY_TITLE...** Changing the title string used for the header on plots.



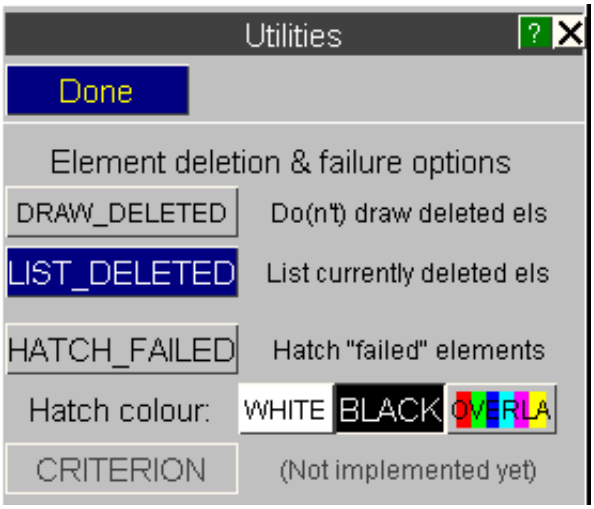
The analysis title taken from the **.PTF** file is normally used for the header on plots, and also for headers in output files.

You can change this by simply typing in a new string, followed by **DONE**. This will take effect the next time you update the plot.

Changes here have no effect on the title stored permanently in the **PTF** file.

Next section.

6.9.3 **FAILURE_OPTIONS...** Controlling and listing deleted and “failed” elements.



This figure shows the **FAILURE_OPTIONS** control panel.

This permits you to control the display of, and to list, elements flagged as "deleted" by LS-DYNA.

It also controls the processing of "failed" elements as diagnosed by D3PLOT.

6.9.3.0 What is the difference between a “deleted” and a “failed” element?

A "deleted" element is one that is flagged in the database file by LS-DYNA as having been deleted in the course of the analysis.

A "failed" element is one that has met some failure criteria within D3PLOT. This capability was added on an experimental basis for casting simulation, and requires the use of a special version of LS-DYNA with modified material models.

Ignore the concept of "failed" elements for now! This capability may be resurrected in the future but, at present, it is dormant.

6.9.3.1 **DRAW_DELETED** Switching the display of deleted elements on/off.

By default deleted elements are not drawn. Once they have been flagged in a plot state as deleted they are removed from the display list altogether. If you turn this switch on they will be drawn.

WARNING: Displaying deleted elements can occasionally crash D3PLOT. Nodes on deleted elements may become unrestrained and, if subject to external force, accelerate off in an arbitrary direction at impossibly high speeds.

Attempting to display facets attached to such nodes, which may have one corner in your model but another somewhere in Ursa Minor, can cause problems for the hidden-line algorithms.

HINT: A better way to visualise deleted elements is to leave this switch ***off***, and instead to use the **DT_DELETION_TIME** component in a solid contoured plot (ie **CT** or **SI** modes). This draws the model with undeformed coordinates, but with deleted elements at the current time shown in a different colour.

6.9.3.2 **LIST_DELETED** Listing deleted elements on the display.

For each element type category a list of elements deleted at the current time is given in a listing box on the display.

(The "failure" options below are included for completeness, but they are not recommended for general use: contact Oasys for more details.)

6.9.3.3 **HATCH_FAILED** Hatching “failed” elements.

When turned on elements which match the current failure criteria are cross-hatched with the current **Hatch colour**.

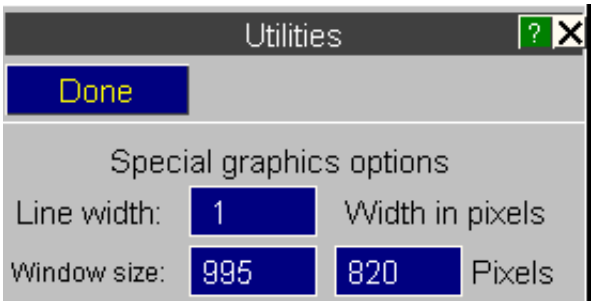
6.9.3.4 **Hatch colour:** Setting the “failed” hatching colour.

At present the options are: **WHITE**, **BLACK**, **OVERLAY**, the default being black.

6.9.3.5 **CRITERION** Setting the “failure” criterion.

This command is not yet implemented in a general form.

6.9.4 **GRAPHICS...** Setting special graphics parameters.



This figure shows the **GRAPHICS** control panel.

This lets you set special attributes of the graphics display.

6.9.4.1 **Line width:** Setting the line width in pixels.

By default all graphics lines are drawn 1 pixel wide, but on hardware that supports it you can increase this to <N> pixels.

A common reason for doing this is to improve the appearance of images being transferred to avi or mpeg. Lines taken from a high resolution computer graphics display often become patchy in movie files, so increasing the line thickness greatly improves their visibility.

You will notice that drawing takes much longer with thick lines: this is because the graphics device has to render more pixels.

6.9.4.2 **Window size:** Setting the graphics window size in pixels

On windows displays it is sometimes convenient to be able to set the graphics window to a specific size in pixels. This command allows you to do that: in screen menu mode the graphics window only is resized, in raw windows mode the whole display window is resized.

The window size is **not** "clamped" by this command: subsequently resizing the window will overwrite this setting.

If you have multiple windows the size will be set for all windows with **`W1 .. Wn`** tabs active.

6.9.5 **DATA_COMPONENTS** Listing the current database contents to screen, and current data components to file.

This command is an aid to debugging the software, so you don't have to know how to use it!

It lists to the screen and to file **`diag.out`** the following table:

D3PLOT version 7.0: Data component summary printout

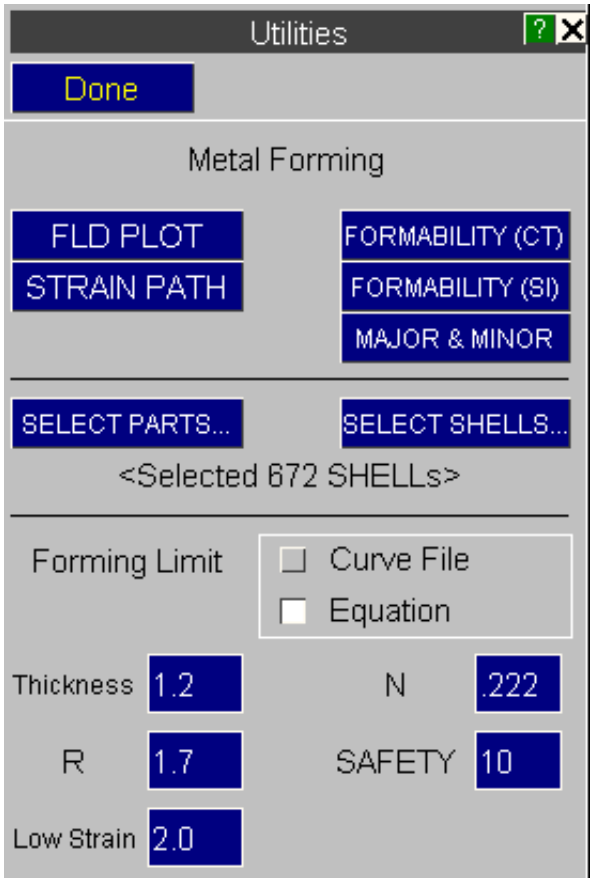
```
====================================
# in database              Solids       Thin shells    Thick shells     Beams
                           ( 16)        ( 4624)        ( 0)             ( 0)
Stress tensor              6 * 1        6 * 3          6 * 3            n/a
Plastic strain             1 * 1        1 * 3          1 * 3            n/a
Strain tensor              6 * 1        6 * 2          6 * 2            n/a
Extra variables            0 * 1        0 * 3          0 * 3            n/a
Forces & moments           n/a          8              n/a              6
Thickness & energy         n/a          4              n/a              n/a
Plastic beam data          n/a          n/a            n/a              0
Deletion flagged           T            T              T                T
```

In addition it writes to file only a listing of the D3PLOT internal data component tables. These are bitwise encoded listings cross-referencing data components against element type and other internal flags. They have no significance to users.

Next section

6.9.6 METAL FORMING

The **METAL FORMING** panel contains a number of special options and plotting modes for use with metal forming analysis. The figure (right) shows the basic **METAL FORMING REFERENCE** control panel.



This panel consist of three sections.

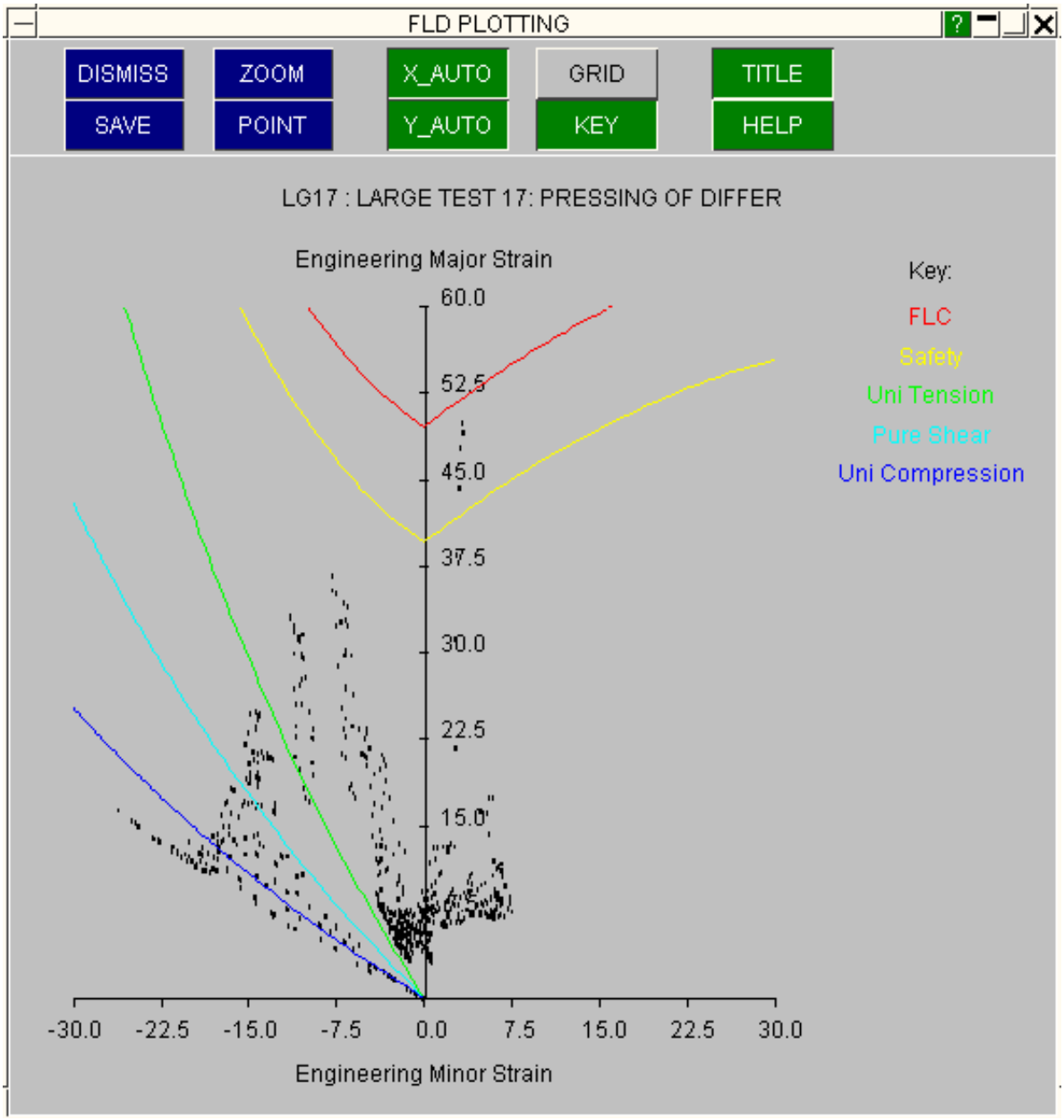
1. The top part of the panel accesses a number of special plotting modes, see Section 6.9.7.1.
2. The middle part of the panel allows entities to be selected for some of the plotting options in the top part of the panel, see Section 6.9.7.2.
3. The bottom part of the panel allows a number of variables to be entered which are used by some of the plotting options in the top part of the panel, see Section 6.9.7.3

6.9.6.1 METAL FORMING plotting options

The top part of the **Metal Forming Panel** contains 5 special plotting options.

FLD PLOT - Forming Limit Diagram

This option will only be available if some entities have been selected for plotting, see Section 6.9.7.2. When selected, this option will produce a graph of **Major Engineering Strain** v **Minor Engineering Strain** at the current analysis time for the selected entities, see the figure below.



In addition to the shell elements that are represented by white dots the graph also contains a number of lines relating to physical and artificial phenomena.

FLC	Forming Limit Curve. This curve represents the point at which the material would fail. This curve may be generated from an equation or read in from a file, see Section 6.9.7.3 for more details.
Safety	This curve represents a safety margin. It is generated automatically by subtracting a constant value from the FLC, see Section 6.9.7.3.

Uniaxial Tension — The uniaxial tension curve is defined by

$$\mathbf{E}_{(major)} = \mathbf{E}_{(minor)} \times -\left(\frac{1+R}{R}\right)$$

Pure Shear — The pure shear curve is defined by

$$\mathbf{E}_{(major)} = -\mathbf{E}_{(minor)}$$

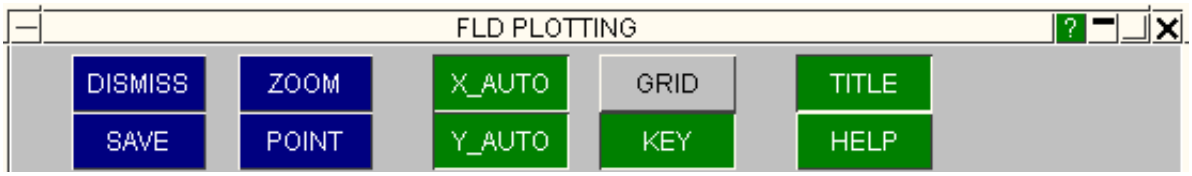
Pure Shear — The uniaxial compression curve is defined by

$$\mathbf{E}_{(major)} = \mathbf{E}_{(minor)} \times -\left(\frac{R}{1+R}\right)$$

where

$\mathbf{E}_{(major)}$, $\mathbf{E}_{(minor)}$	True major/minor strains
R	Normal Anisotropy Parameter

Graph Options



DISMISS Closes the FLD plotting window.

SAVE Save the points and the curve data to a curve file for reading into Oasys T/HIS.

ZOOM Uses the cursor to zoom in on a rectangular area of the graph, and enlarges the area to fill the whole window.

POINT Select a point on the graph. The ID of the nearest shell to the point is then written out to the dialogue area and the shell is labelled in the graphics window if it is visible.

X_AUTO Resets the X scaling to automatic.

Y_AUTO Resets the Y scaling to automatic.

GRID Toggles a grid at the tick mark intervals on/off (Default ***off***).

KEY Toggles the curve "Key" on/off (Default ***off***).

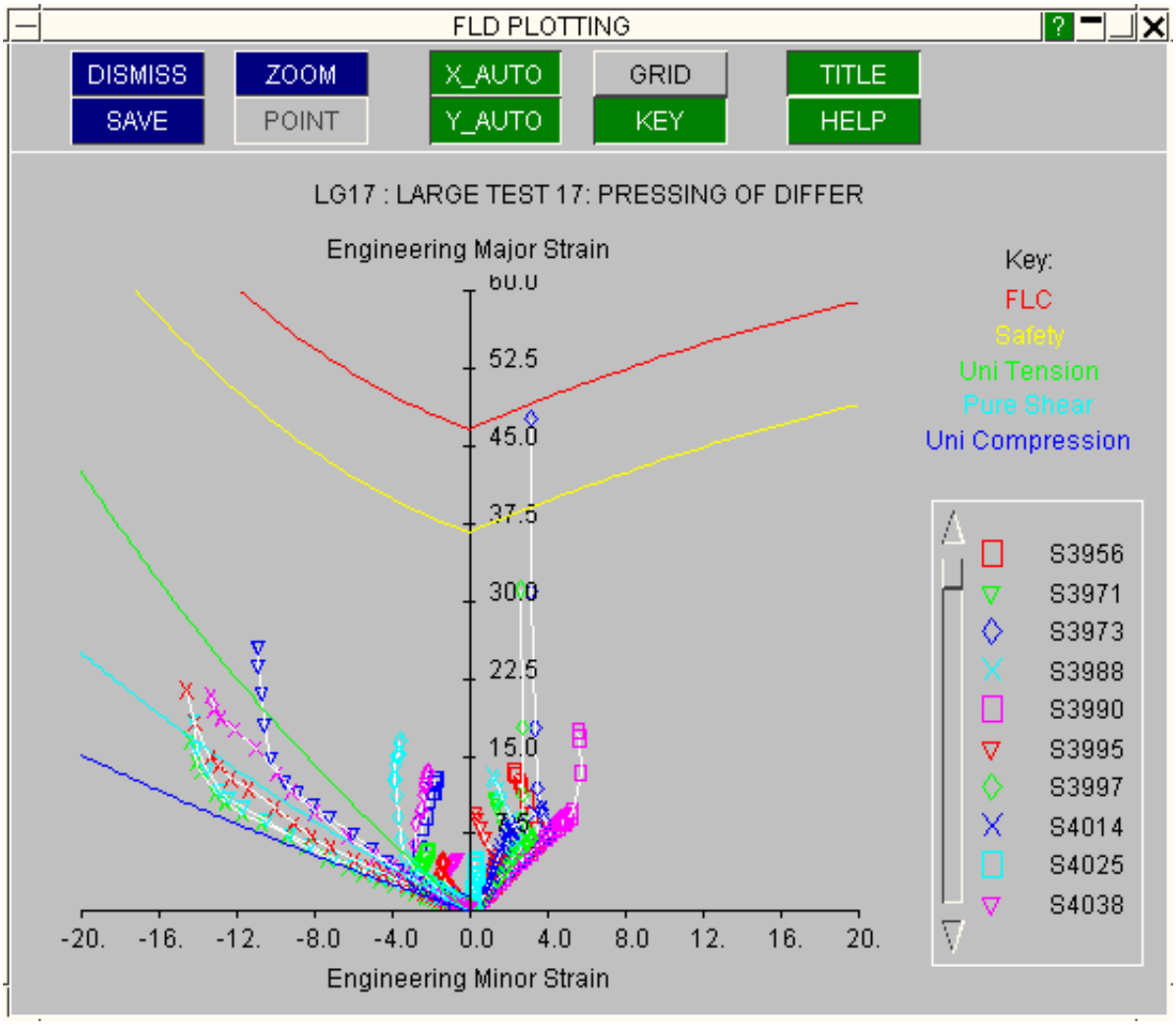
TITLE Toggles the analysis title on/off (Default ***on***).

HELP Gives on-line help.

STRAIN PATH

The **STRAIN PATH** option allows the user to plot the **Major Engineering Strain** v **Minor Engineering Strain** path for the selected entities, see Section 6.9.7.2. This produces a plot of **Major Engineering Strain** v **Minor Engineering Strain** at every plot-state, see the following figure. This option is useful as it shows the strain-route taken by an element. For example, if the material properties defined by the user do not allow failure then this plot may help to highlight elements that appear to be okay at the end of the analysis but are likely to have failed during the analysis if failure criteria had been included in the material model.

If the selected entities are shells then D3PLOT will display the first 20 shells that have been selected. If a material has been selected then D3PLOT will only display the first 20 shells in that material.



FORMABILITY (CT & SI)

The **FORMABILITY** options produce either a continuous tone (**CT**) or shaded image (**SI**) contour plot indicating the formability of the product, see the two figures below. These options will only be available if Forming Limit Curve (FLC) has either been read in or generated within D3PLOT, see Section 6.9.7.3.

The formability of each element is determined by its position on the FLD as follows.

FAILED	Above the FLC.
MARGINAL	Between the FLC and the safety curve.
SAFE STRETCH	+ve minor strain and below the safety curve.
SAFE PLANE STRAIN	Below safety curve, abs[minor strain] < 5%[major strain]

SAFE DRAW -ve minor strain, between safety curve and pure shear curve.

MARGINAL WRINKLING Between pure shear curve and uniaxial compression curve.

STRONG WRINKLING Below uniaxial compression curve.

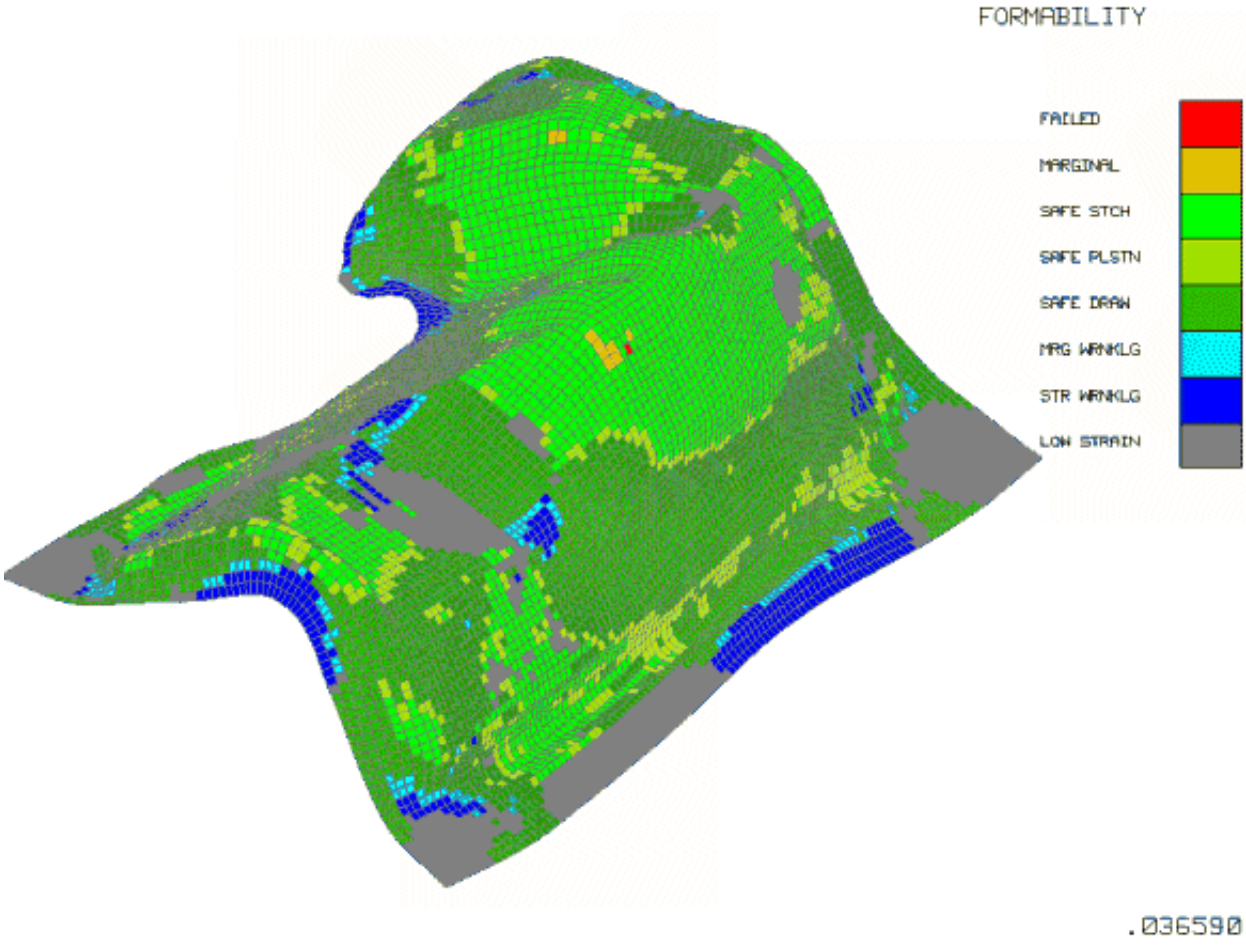
LOW STRAIN $\sqrt{(E^2_{(major)} + E^2_{(minor)})} \leq Low\ Strain\ Value$

These options automatically set the following items before the plot is generated.

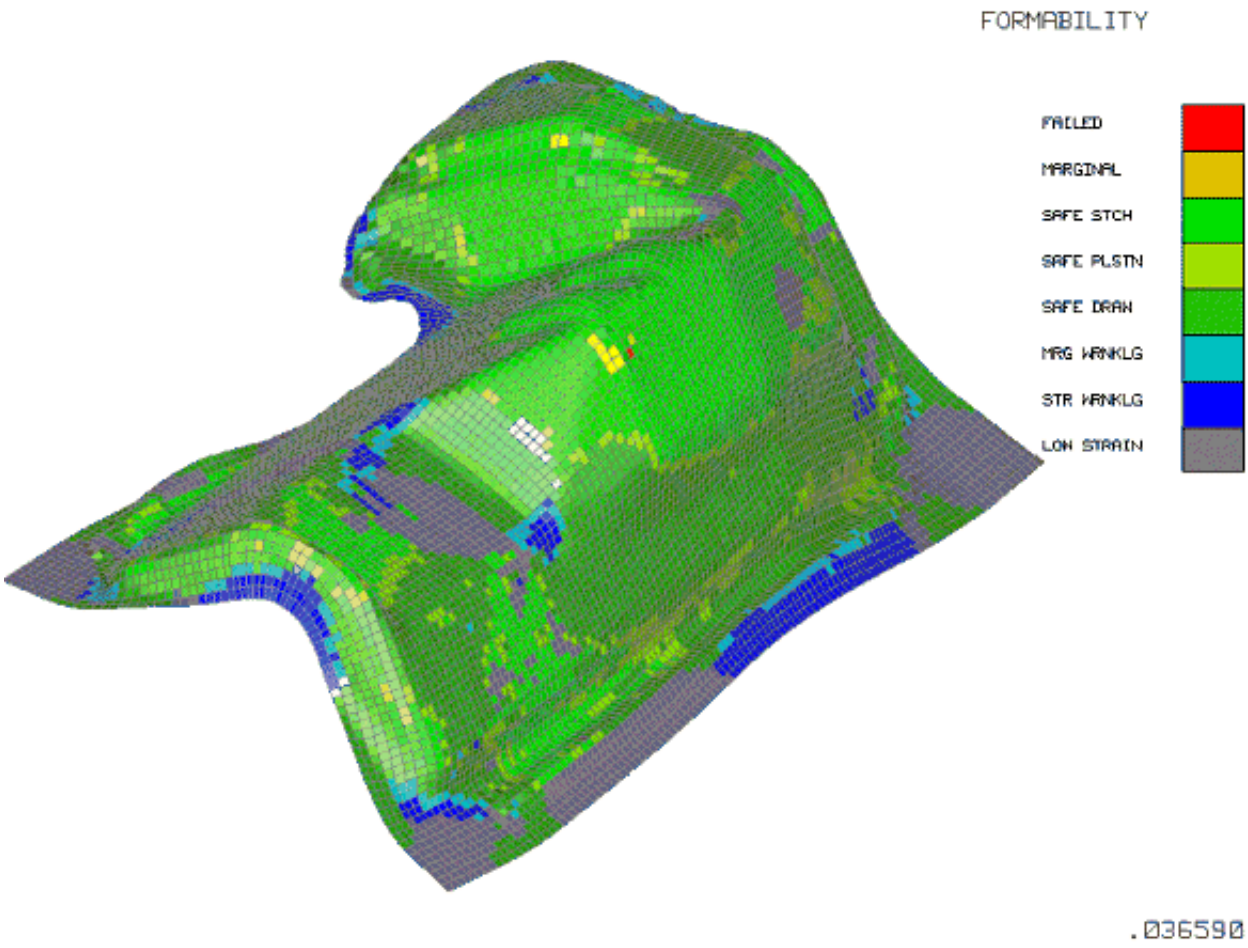
- 8 Contour levels and colours (see Section 4.3)
- No element averaging (see Section 4.2.4.5)

After the plot is completed these values are reset to their previous settings.

FORMABILITY (CT) Plot

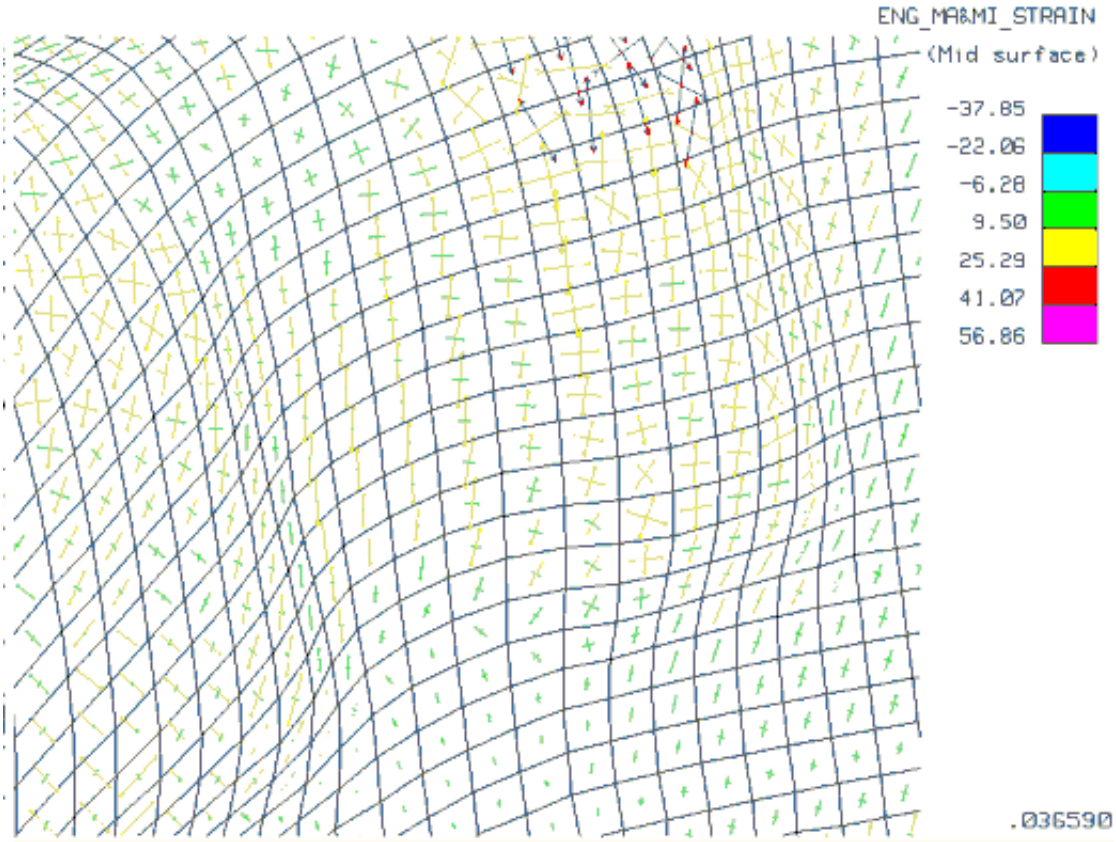


FORMABILITY (SI) PLOT



MAJOR & MINOR

The **MAJOR & MINOR** option produces a vector plot of the MAJOR and MINOR engineering strains, see the figure below, where the Major Engineering strains are shown with normal arrow heads and the Minor Engineering strains are shown with inverted arrow heads. This can only be plotted if the strain tensor has been output to the PTF files for shell elements.



The following items may be modified before the plot is generated by changing the values in the Principal Stress/ Strain menu, see Section 4.2.7.

- Number of contour levels
- Contour colours
- Length of the vector arrows
- Shell surface (top, middle and bottom)

The engineering strains are calculated from the shell element strain tensor using the following equations. These calculations ignore any through thickness strains.

$$Major\ Engineering\ Strain = (e^{\varepsilon_{major}} - 1) * 100$$

$$Minor\ Engineering\ Strain = (e^{\varepsilon_{minor}} - 1) * 100$$

where

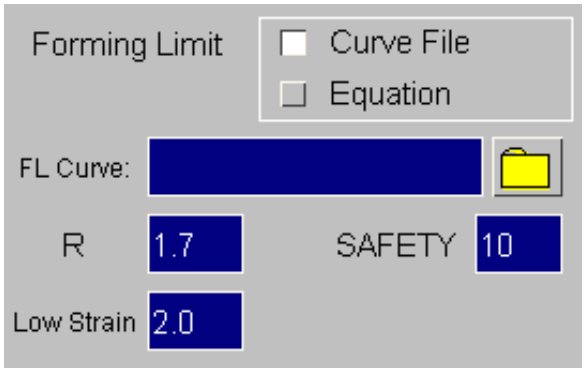
$$\varepsilon_{major} = P_1 + P_2$$
$$\varepsilon_{minor} = P_1 - P_2$$
$$P_1 = \frac{\varepsilon_{xx} + \varepsilon_{yy}}{2}$$
$$P_2 = \sqrt{\left(\frac{\varepsilon_{xx} - \varepsilon_{yy}}{2}\right) + \varepsilon_{xy}^2}$$

6.9.6.2 **METAL FORMING** entity selection

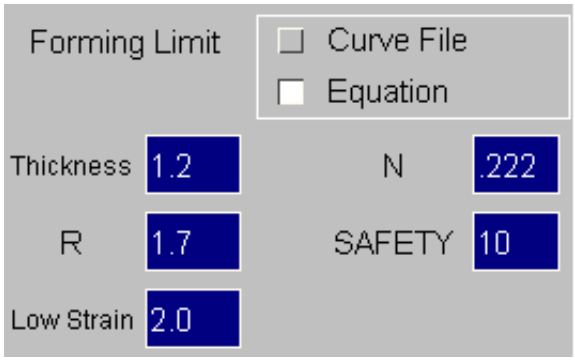
The metal forming options only apply to 3 and 4 noded shell elements and therefore only shells or shell materials can be selected. The panel also shows how many shells have been selected.

6.9.6.3 **METAL FORMING** data values

The bottom section of the **METAL FORMING** panel allows a Forming Limit Curve (FLC) to be read in from a curve file (see figure right), or to be generated by D3PLOT from an equation (see figure below right). The curve file name is entered in the box. The parameters **R**, **Safety** and **Low Strain** are added to generate the other curves.



If the FLC is generated from an equation the following parameters are entered in the boxes: **Thickness**, **N**, **R**, **Safety**, and **Low Strain**.



The FLD curves are then generated according to an empirical formula which is generally applicable for steels as follows:

```
If E[minor] < 0        E[major] = FLD + E[minor] * ( 0.042 *
If E[minor] > 0        E[minor]+0.627 )
                       E[major] = FLD + E[minor] * (-0.0086 *
                       E[minor]+0.785)
```

Where **FLD** = (N/0.2116) * (23.36 + 14.042*min(T, 3.0))

R = Normal Anisotropy Parameter

N = Work Hardening Parameter

T = Material Thickness (mm)

Other data values

SAFETY Forming Limit Safety Margin.

LOW STRAIN Strains are ignored if $\sqrt{(E_{(major)}^2 + E_{(minor)}^2)} \leq Low\ Strain$

The safety curve on the **FLD** and **STRAIN PATH** plots is automatically generated by subtracting the SAFETY value from the FLC.

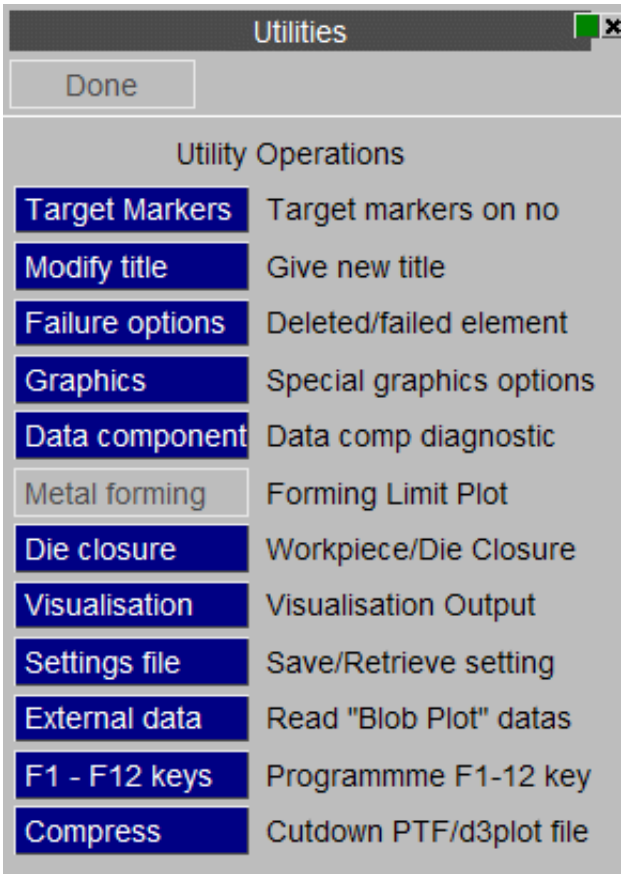
Next section

6.9.7 **UTILITIES**, **DIE_CLOSURE**

The "Die-Closure" command set, found in the **UTILITIES** menu creates and controls the special **CLOSURE** data component.

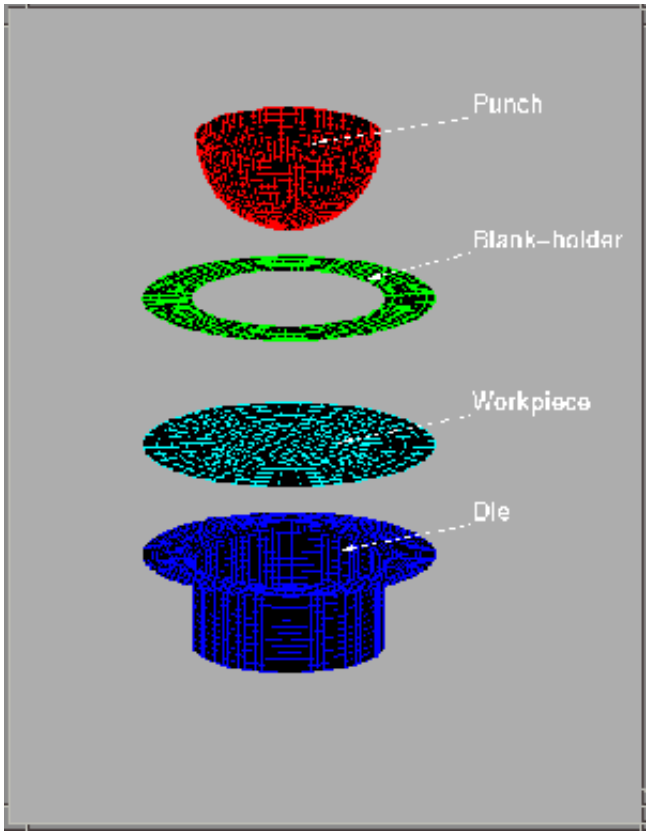
This data component calculates the distance remaining before a "workpiece" makes contact with its "die".

The terminology implies metal-forming, and the most common use of this will be to determine how much further a deformable blank has to be pressed in order to achieve its desired shape. But any problem requiring the distance between two pieces of 2D/3D structure to be calculated can be solved here, and it has application to many cases in which "distance to contact" needs to be known.

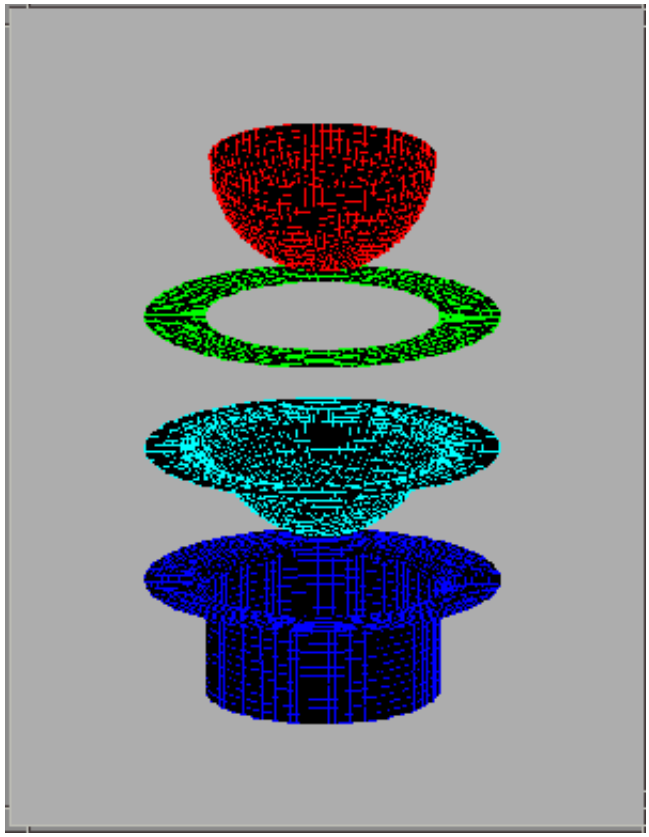


6.9.7.1 Defining the problem

The two figures below show a simple pressing problem, parts exploded, in which the workpiece (blue) is pressed by a punch into a cup shape. We need to know how far the punch (red) must be pushed down to achieve the required final shape.

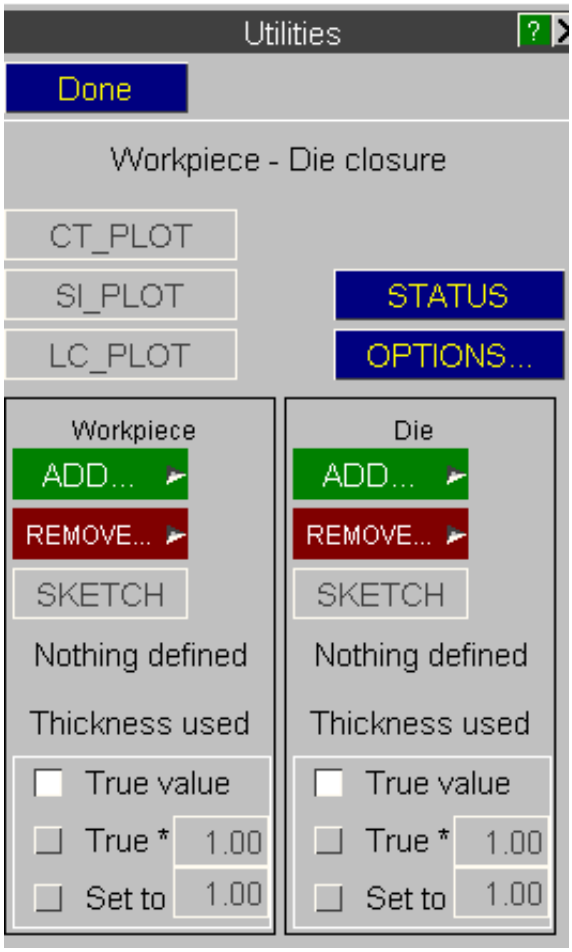


Intial problem layout


Final shape

6.9.7.2 Defining the "Workpiece" and "Die" sides.

Closure distance is computed between nodes on the "workpiece" and element faces (facets) on the "die". Therefore it is necessary to tell D3PLOT what each side actually is, as in LS-DYNA contacts.



Once a type has been chosen the standard D3PLOT selection menu is used to define the actual elements or parts.

Each side may be any mixture of 2D and/or 3D elements: solids, shells, and thick shells, although in most cases (as here) shells will be used. Sides may be defined by element and/or part, and may have elements added and removed at will, and elements may be deformable or rigid. Definition by part has advantages in adaptively remeshed cases, as in this example, since the part definition is "remembered" across remeshes and the extra elements are auto-matically included in the relevant side.

Elements may not exist on both sides at once, and adding an element to one side that already exists in the other will cause it to be removed from the older definition. This cross-check is automatic. The definition panel is updated immediately to show how many elements there are on each side, and you can **SKETCH** each side at any time to confirm your choice.

6.9.7.3 Plotting closure

Once there is at least one element on each side the **CT_PLOT**, **SI_PLOT** and **LC_PLOT** buttons will be made "live", and these will immediately compute the closure values, set the programme-wide data component to **CLOSURE**, and perform a plot in the relevant mode (continuous-tone, shaded image, line contours).

You can also select the **CLOSURE** data component explicitly from the **DISPLACEMENTS...** data category, and plot it using main menu commands. It is also a scalar (nodal) data component that can be used in **WRITE** and **XY_PLOT** contexts.

6.9.7.4 How the “closure” calculation works, and the output it produces.

It is necessary to calculate the distance from each workpiece node to the nearest facet on the die. This is done by determining which facet a node will be projected onto when contact is made, and computing to the vector distance (less element thicknesses) between node and facet. This is very similar to the ***CONTACT_NODES_TO_SURFACE** problem in LS-DYNA, except that D3PLOT must consider nodes at some distance from, as well as in contact with, the die surface.

For nodes close to or in contact with a facet this is a simple calculation, but when a node is quite a long way from the

die problems of ambiguity ("Which facet will I ultimately make contact with?") and complexity ("I need to be tested against many distant facets") arise.

To get round these problems, and to achieve a reasonably short computation time, a bucket-sorting process is used:

- The volume of space around the workpiece, plus a margin, is calculated and then sub-divided by (x,y,z) coordinates into a 3D matrix of "buckets". Each bucket is a smaller, rectilinear volume of space.
- Each workpiece node is assigned to a bucket, based on its coordinate. Die facets may lie in one or more buckets, or none at all if very distant.
- For each die facet closure is only calculated for workpiece nodes that exist in the same or immediately adjacent bucket(s) as the facet.

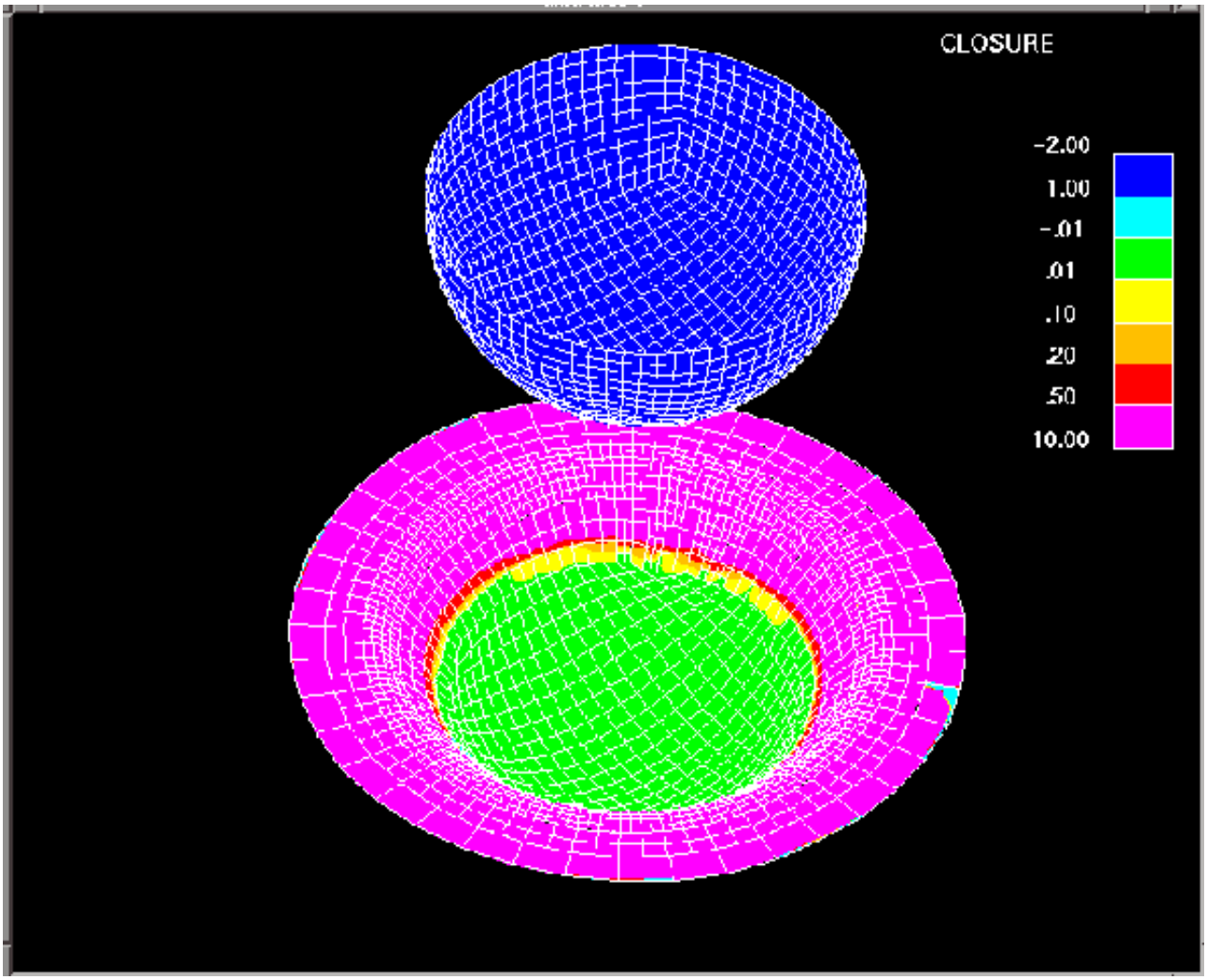
D3PLOT uses a 10x10x10 (x,y,z) array of buckets, and chooses default dimensions such that closure values up to about 20% of the longest workpiece dimension will be calculated. Thus node closures will fall into one of four categories:

Node location	Meaning	Value
In Contact	The workpiece node lies on or behind the die facet surface. (The penalty algorithm in LS-DYNA means that a node may penetrate an element.)	0.0
Near to contact	The node is reasonably close to a facet, and a closure distance can be calculated.	> 0.0
Uncomputed	The node is too far from the die for a closure distance to be computed, or has fallen into a "tunnel" between facets (see below).	-1.0 [1]
Uninvolved	For nodes not in the workpiece, so not involved in the closure calculation at all.	-2.0 [1]

Closure categories and associated values

Note 1: These values are defaults which can be changed, see **OPTIONS...** below.

6.9.7.5 Typical closure output



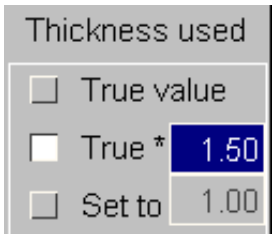
This figure illustrates the final state of the example shown earlier. Here the "die" side is actually the punch, since that is the shape against which we wish to measure closure.

Note that the contour levels have been chosen explicitly, with unequal intervals, to illustrate the following:

- The centre of the workpiece is fully in contact: green area, value = 0.0.
- There is a zone of progressive opening roughly half way up the walls, where a gap between workpiece and punch is opening up: yellow (0.01) to red (0.5).
- The rest of the workpiece has a gap of 10mm or more (purple).
- All punch (ie the "die" side) nodes, for which no closure values are calculated, are blue, having the value of -2.0.
- There is a small patch of "uncomputed" nodes (light blue, = -1.0) at about 4 o'clock on the workpiece. The reasons for this are explained later.

6.9.7.6 Controlling the die-closure calculation process

Controlling the thickness used.



By default the true thickness of both workpiece and die is used when calculating closure. "True" thickness depends on the element type:

(Thin) shells	The current element thickness as reported in the **`.ptf`** file. The centreline +/- half this value is used.
Solids and Tk. Shells	Zero. (The nodes are assumed to lie on the face surface.)

For workpiece nodes the average thickness of all elements meeting at the node is used, for die facets the actual value is used.

For each side individually you can apply a factor to the "true" value, or override it with an explicit value. (You may wish to do this if you have used artificial thicknesses during the calculation, since this will affect contact geometry.)

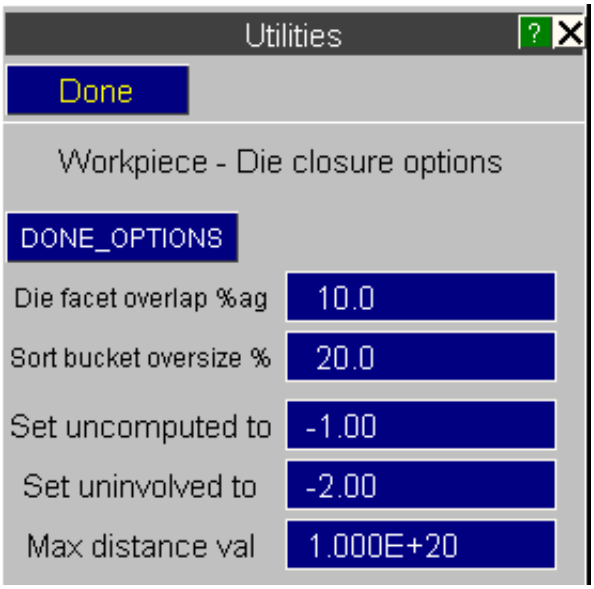
OPTIONS... Controlling calculation parameters
The default parameters chosen by D3PLOT should be satisfactory in most cases, but there are situations when you may wish to alter them.

The **OPTIONS...** command gives access to the following adjustable parameters.

"Die facet overlap %age" controls the extent to which die facets are artificially enlarged (in the in-plane direction) for the purposes of checking workpiece node projection.

This is important where the die surface is convex with respect to the workpiece, since it helps to prevent workpiece nodes falling into the "tunnels" between projected facet volumes.

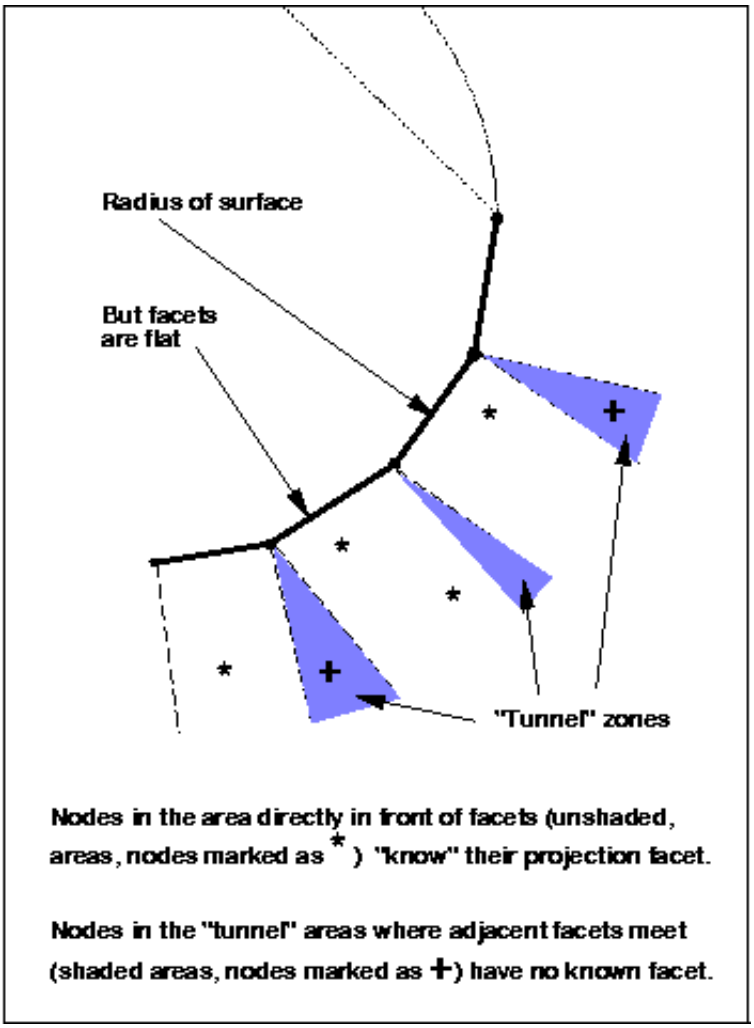
This can be illustrated as follows:



Using the Die facet overlap %age value to prevent nodes being lost on convex surfaces

When workpiece nodes are being projected onto a convex die surface problems can arise when nodes lie in the dead zones ("tunnels") where facets meet.
This is illustrated here: nodes marked * lie in the areas opposite die facets, and so "know" which facet they are likely to be projected onto.

Nodes marked + in the shaded tunnel areas don't "know" which facet they will be projected onto, and so no closure value is calculated for them.
Clearly this problem becomes more acute as the distance of a workpiece node from the die increases, and is the main reason why closure values for such distant nodes may appear randomly to be classified as "uncomputed", and given a value of -1.0.

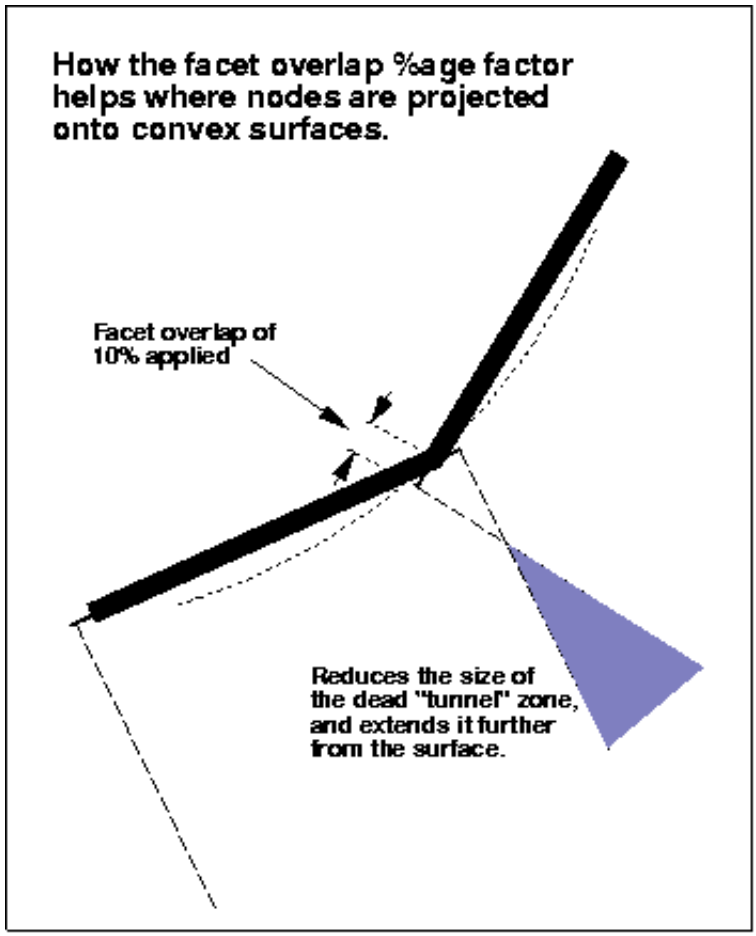

Radius of surface
But facets are flat
"Tunnel" zones
Nodes in the area directly in front of facets (unshaded, areas, nodes marked as *) "know" their projection facet.
Nodes in the "tunnel" areas where adjacent facets meet (shaded areas, nodes marked as +) have no known facet.

The situation can be improved by artificially increasing the width of die facets so that they overlap, using the "Die facet overlap %age" factor.

This is illustrated here: a value of 10% (the default) has been used, and this shows how the "tunnel" dead zone has been made smaller, and also extended further away from the die surface.

Clearly larger values will lead to fewer nodes being lost in "tunnels", but this must be balanced against the fact that it may lead to invalid closure values being calculated when nodes are projected onto the wrong facets because these have become artifically extended.

This is a case for engineering judgement.


How the facet overlap %age factor helps where nodes are projected onto convex surfaces.
Facet overlap of 10% applied
Reduces the size of the dead "tunnel" zone, and extends it further from the surface.

Using the Sort Bucket oversize %age to include distant die facets

The volume of space used for bucket sorting is based on the dimensions of the workpiece, not the die. Therefore if the die is much larger, or some considerable distance away, then there is a good chance that many facets on it will lie outside the bucket volume, and this means that they won't be considered when computing closure.

Increasing the Sort bucket oversize %age value from its default of 20% may include these ...

BUT:

- Closure values calculated from distant facets are likely to be unreliable, since by the time the workpiece and die meet at that point their respective structures will probably have been deformed so much that contact will actually occur at some other location.
- Increasing the bucket volume will increase the number of nodes and facets in each bucket. The time taken to compute closure rises as a function of ($N_{Workpiece} * N_{Facet}$) **in each bucket, and these values increase as a function of bucket volume, ie linear dimension cubed. So increasing the bucket size can lead to a rapid rise in the time taken to compute closure, in the worst case by a fourth power.**

Or, put another way, you need good reasons to increase this value!

Controlling the values used for uncomputed, uninvolved and distant nodes.

These values do not affect the calculation at all, only the values assigned to special cases.

Set uncomputed to:	Is the value assigned to nodes on the workpiece for which no closure value can be computed. By default this is -1.0, but you can choose any value - although you should avoid values that might be confused with valid results (ie zero or small +ve numbers).
Set uninvolved to:	This is the value assigned to all nodes in the model that are not part of the workpiece. The default is -2.0, but again you can choose any sensible value.

Max distance val: Nodes with computed closure values greater than this value have them reset to the "uncomputed" value. By default this is +1.0e20, ie no nodes will ever fall into this category, but you can set it to a sensible upper-bound value.

The advantage of using negative values for uncomputed and uninvolved nodes is that they will never be valid closure distances, and so can be isolated during contouring by judicious choice of contour bounds, or excluded from plots altogether with the **Limiting Values** option in the **CONTOUR** menu.

6.9.7.7 Closure calculation time

Despite bucket-sorting the closure calculation is still numerically intensive, and can take some time to perform. Therefore the following strategy is adopted:

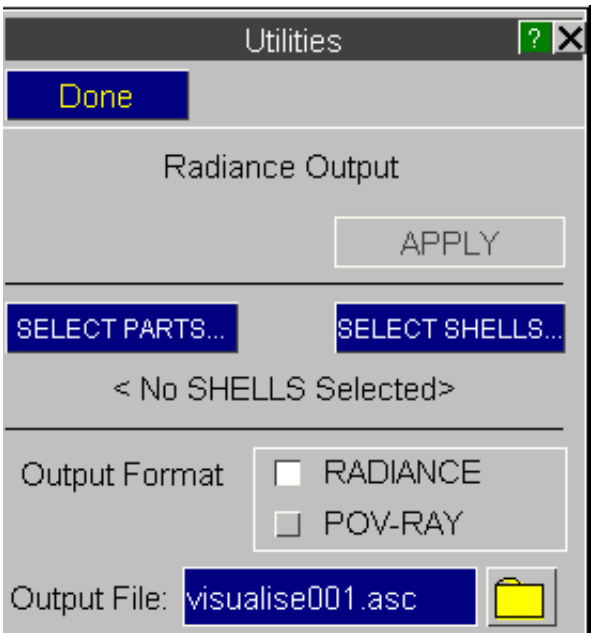
- The closure calculation is only carried out when actually required. For example when a plot is requested, or scalar data extracted
- Once calculated the results are stored, so that subsequent plots, changes of contour values, etc, are fast as no recalculation is required.

The closure calculation has to be repeated when:

- Either the workpiece or die contents are changed, by adding or removing elements.
- Any closure parameter (thickness, bucket size, etc) is changed.
- A new complete state is read in.
- A velocity arrow or contact force vector plot is carried out. (Since the temporary storage space used is overwritten by these two plotting modes.)

While closure is being calculated a "progress" bar is written to the dialogue box stating how far, in %age terms, the calculation has gone. You can use **STOP** to terminate it prematurely at any time.

6.9.8 VISUALISATION



The figure (right) shows the **VISUALISATION** control panel. This option allows input files for ray tracing codes to be generated.

Output files may be written in a format suitable for either **RADIANCE** or **POV-RAY**.

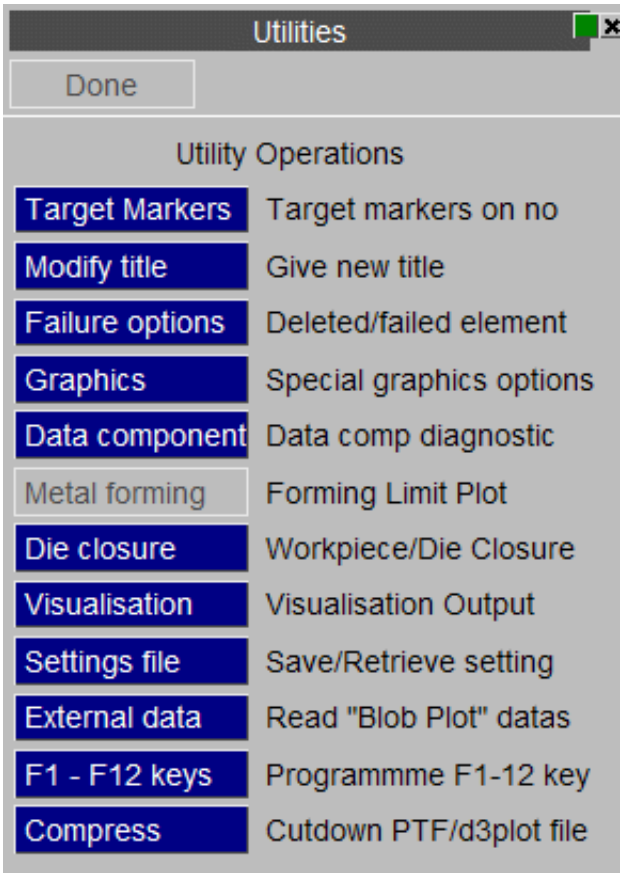
Notes

1) Only shell elements/materials may be selected for output.

2) ASCII files generated using this option can become very large due to the input formats required by **POV-RAY** and **RADIANCE**.

Next section

6.9.9 UTILITIES, SETTINGS_FILE



The "settings" file (extension **.set**) stores programme and window status information.

Its purpose is to allow you to define a screen layout that is model-independent, and to revert to that layout with minimum effort.

The settings file stores the number of graphics windows, and for each window:

- The current view.
- The background, text and hidden infill colours.
- The current **ENTITY** and **LABEL** settings.
- The current data component information and plotting mode.
- The current contouring settings and colours.
- Any cut-section data.
- Any fixed node, shift deformed and reference data.
- Lighting and shading information.
- Most **DISPLAY_OPTIONS** settings.
- Other miscellaneous data.

Reading in a settings file will optionally open further windows if the number currently present is less than that in the file, then apply the stored settings to all the windows in the display.

The settings file (**.set**) stores ***Programme***, not *Model*, information.

Almost all the values in the settings file are model-independent, and can be applied to any analysis. The exceptions are:

The current view.	Both the direction cosines (the view direction) and the current centre, scale and perspective distance are stored. If applied to a model with significantly different dimensions it may be necessary to **AC** (autoscale) after reading the settings file in order to make the model visible.
Nodes used in **Fixed node**, **Shift Deformed** and **Relative Values**.	If any of these nodes are defined they are converted to their external labels when written to the settings files. When the file is read back in an attempt is made to match the external label in the current model. If this is successful (for all nodes in a context where > 1 are required) that mode is restored, otherwise that mode is switched off.
Nodes used in **Cut_Sections**, and "**Cut follows nodes**"	Likewise nodes used to define a cut-section are written using external labels. When reread an attempt is made to match these in the current model. For cut section ***definitions*** it doesn't matter if the node(s) are not matched, since they are only used to derive section geometry and can be dispensed with once this has been computed. However if "**Cut follows nodes**" has been selected, but any of the nodes cannot be matched, this mode will be switched off.

The Properties (**.prp**) file stores *Model-specific* information.

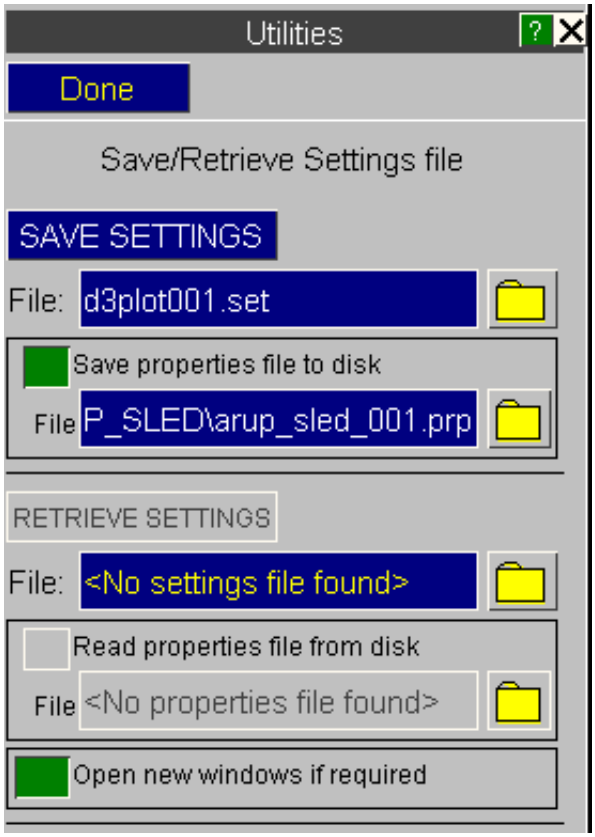
Information about models: item colour, transparency, brightness, overlay, labelling, etc is stored in the "properties" file. More details about this file are given under **PROPS** in section 4.2.2.4.

Properties files are written using external labels, so they can be applied to different models. Where an external label is not matched in the current model, or no counterpart to a current item is found in the properties file, no action is taken. Therefore properties files can be used to restore model settings to broadly similar analyses.

However the Settings and Properties files are closely related.

Because these files are so closely related in terms of what they do it makes sense to process them together, and the **SETTINGS_FILE** panel allows you to read and write both types.

6.9.9.1 The **SETTINGS FILE** Panel



SAVE_SETTINGS

Stores the settings of all current windows in the specified "**.set**" files.

The default filename "**d3plot*nnn*.set**" is recommended, although any filename may be used.

If **Save properties file to disk** is selected then the specified properties file is also written. Any filename may be used, but the extension "**.prp**" is recommended.

RETRIEVE_SETTINGS

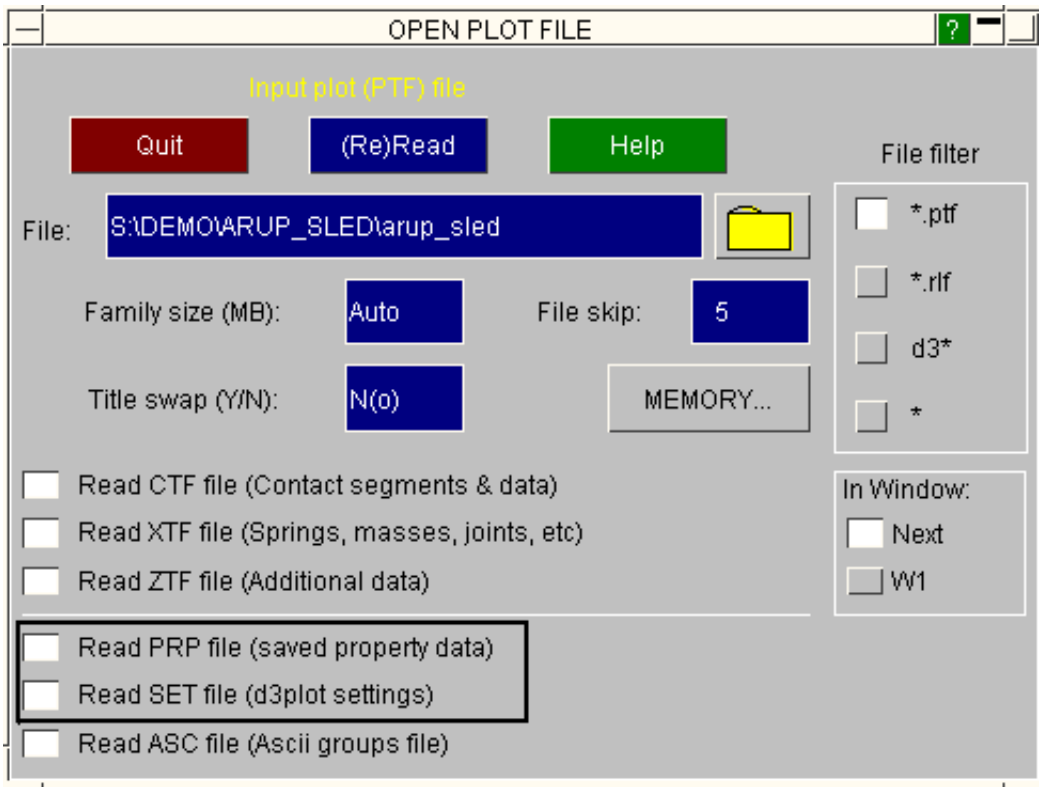
Reads in the specified settings file. If several are found the most recent is shown.

If **Read properties file from disk** is selected the specified properties file will be read. In this example no such file has been found, so this option is not available.

If **Open new windows if required** is selected then, if the settings file contains information for more windows than are current, additional windows up to this limit will be opened. Otherwise settings will only be applied to existing windows.

If more windows are current than are defined in the settings file, then the stored values for window #1 will be applied to the excess ones.

6.9.9.2 Reading Settings and Properties files with a new model



On the **New Model** panel you can choose to read any Settings or Properties files present after you have read in the model.

The default filenames used will be:

The most recent Properties file in the model directory of the name:

`<filename>_nnn.prp`

The most recent Settings file in the current directory of the name:

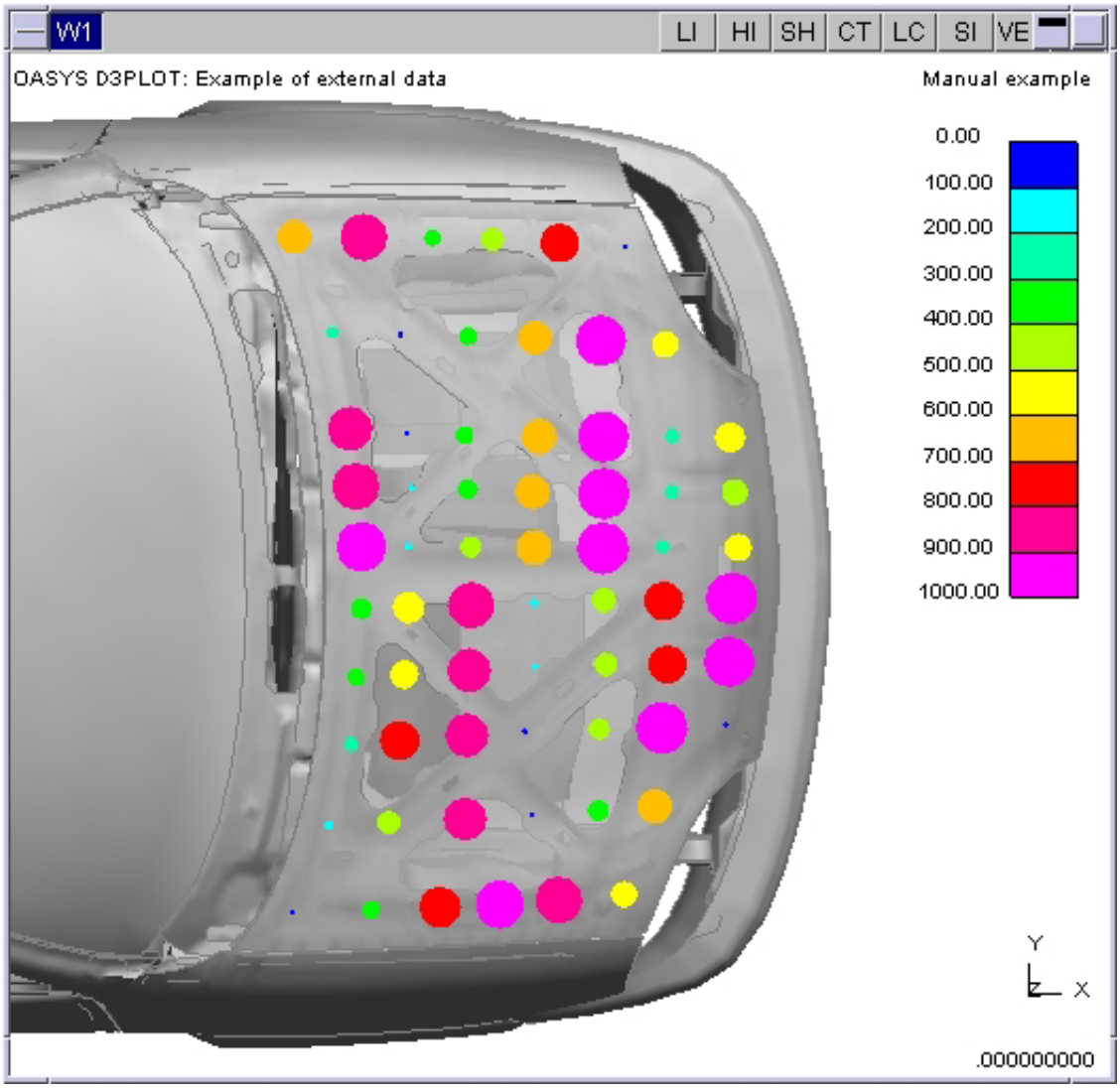
`d3plotnnn.set`

These will be applied as if they had been read in explicitly above using **RETRIEVE_SETTINGS**.

6.9.10 External Data

"External" data is arbitrary data at three-dimensional points, generated externally, that D3PLOT can superimpose on the current image in a variety of ways.

The following - artificial - example shows notional head-impact values on a car bonnet, but any data from any source can be used.
(External data plots are sometimes referred to as "Blob plots", for reasons that are obvious from the plot below!)



6.9.10.1 How External Data plotting works.

An "external data" file is generated (externally) by the user, by any means, that contains some or all of the following attributes:

- Arbitrary data values at three-dimensional coordinates in space
- Contour band values and colours
- Information about symbol size and labelling
- Lines used to demarcate regions

D3PLOT reads this file, supplying defaults for information that is not specified, and displays the results on top of the current image. This display can be turned on or off at will, and modified at any time.

External data is stored on a "per model" basis, so if you have multiple models data has to be read separately for each model and its visual attributes may also be controlled separately.

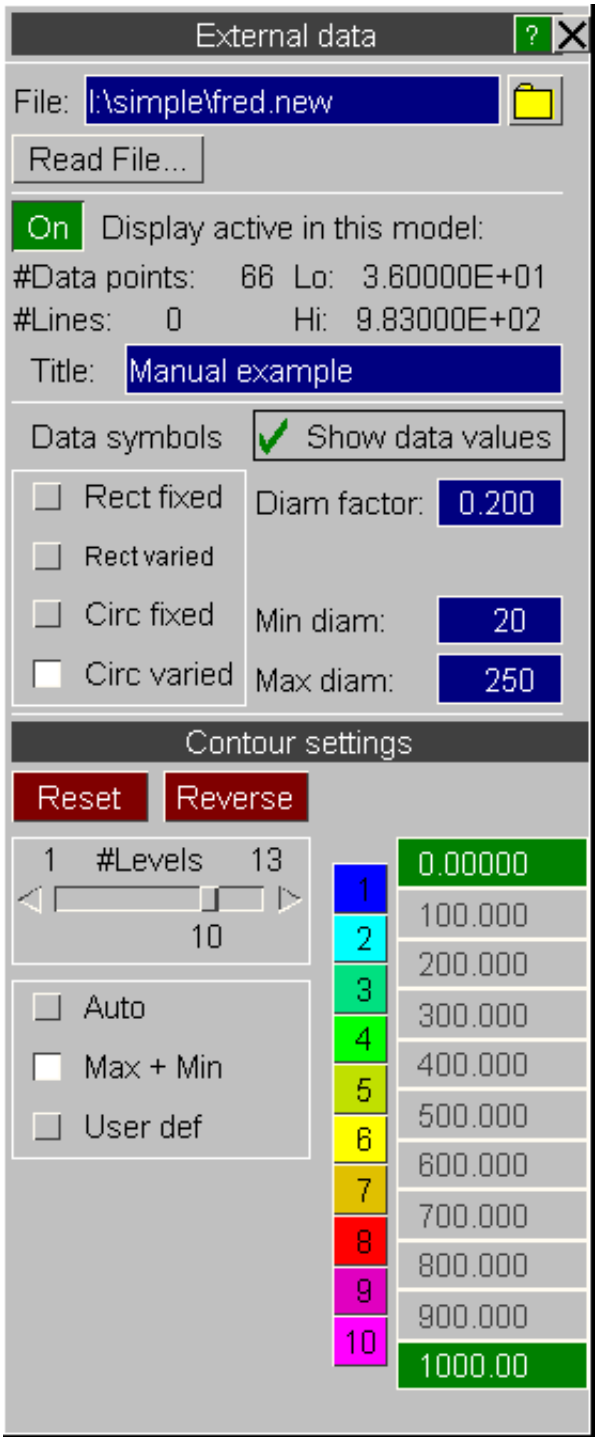
6.9.10.2 The format of an "External Data" file.

External data files are free format ASCII files in which data can appear in any order (unless stated otherwise below). They may contain any or all of the following information:

Header	Data type	Data format example
$ xxx and blank lines	Lines starting with "**$**" and blank lines are treated as comment lines, they can appear anywhere in the file and are ignored	**$ This is a comment line**
name <*data name*>	A title of up to 30 characters	**name This is a demonstration example** Will read the title "This is a demonstration example"
automatic <*#bands*>.	Denotes automatic contours to be used with <#bands> levels where 1 <= #bands <= 13 Contour bounds are determined by scanning the data once read.	**automatic 10** Will use automatic contour levels with 10 bands
levels <*#levels*> **<*L1*> <*rr*> <*gg*> <*bb*>** **<*L2*> <*rr*> <*gg*> <*bb*>** to #levels lines	Denotes manually specified contours with <#levels> values where: 2 <= #levels <= 14 Each level has a value <Ln> followed by Red Green Blue colour components, each component being in the range 0 - 100. Colour <i> spans the band <i> to <i+1>	**levels 6** **0 0 0 100** **5 0 100 100** **10 0 100 0** **15 100 100 0** **20 100 0 0** **25 100 0 100** Sets up 6 levels, ie 5 bands, from blue to magenta
Note on contour levels / bands: • Commands "**automatic**" and "**levels**" are mutually exclusive, you cannot have both. • If neither is defined then the default is "**automatic**" with the same number of levels as currently used for normal contouring. • In the "**levels**" case you must follow the levels command with the correct number of lines of data.		
circle_f <*diam*> **circle_v <*factor*> <*min_dia*> <*max_dia*>** **rect_f <*width*> <*height*>** **rect_v <*fac_w*> <*fac_h*> <*min_dim*> <*max_dim*>**	Only one of these options may be used. • **circle_f** is a fixed diameter circle • **circle_v** is a variable diameter circle, with diameter set to <**factor** * **data**> • **rect_f** is a fixed size rectangle • **rect_v** is a variable size rectangle of width <**fac_w** * **data**> and height <**fac_h** * **data**> In both "variable" cases there is limiting minimum and maximum size for the symbol. If nothing is defined the default is a fixed size rectangle of 20 x 20 units.	**circle_v 0.1 30 250** Would set a variable diameter circle based on 0.1 * the incoming data value, and subject to a minimum size of 30 units and a maximum of 250 units. **rect_v 0.2 0.1 100 300** Would set a variable sized rectangle, width = 0.2 * data, height = 0.1 * data, with a minimum size of 100 units and a maximum of 300 units. Screen units are based on the scaled screen unit of 4096 units across the window width.
show_value <true/false>	Whether or not the data value is drawn as a label on top of each point's symbol. The default is **false**.	**show_value true** Would cause values to be drawn.

data <x,y,z> <value>	Data point values. Each point must have an (x,y,z) coordinate followed by a value. The value may be floating point or integer. Any number of data values may be input, by default none is defined.	**data 10.1 20.2 30.3 500.0** Would make a point at position (10.1, 20.2, 30.3) with a value of 500.0.
beam <x1,y1,z1> <x2,y2,z2> <colour>	Defines a "beam" (really just a line) from (x1,y1,z1) to (x2,y2,z2) in colour number <colour>. At present colours are hard-wired as follows: 1. Red, thin line 2. Green, medium width line 3. Blue, thick line (This feature is a crude solution and will probably develop in the future.)	**beam 1.0 1.0 1.0 2.0 2.0 2.0 2** Would drawn a medium width green line from (1,1,1) to (2,2,2)

6.9.10.3 Reading and controlling External Data files.



Utilities, External Data maps the panel shown here.

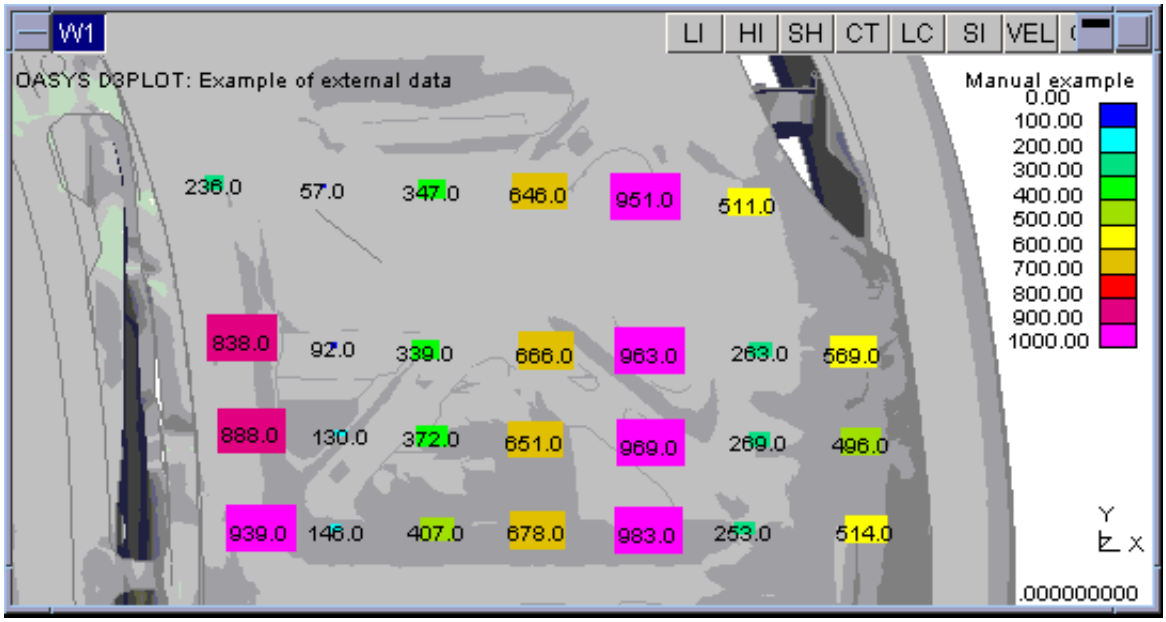
Use the **File:** prompt to define the external data filename, then **Read File...** to read it in and store it in memory.

Thereafter the key parameters described above (contour bands, symbol type and size, data value display) will be used to initialise this panel, but you can change them interactively at will.

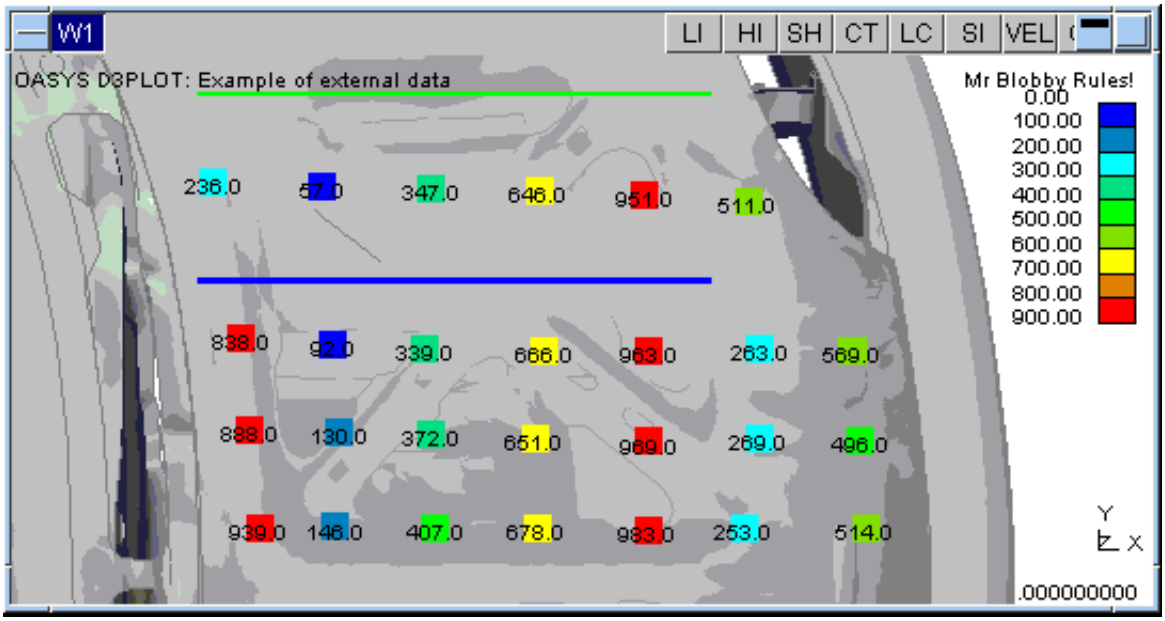
The turning off/on of display of external data on plots is controlled by **Display active in this model.**

Some examples. Note that displaying the underlying model in grey usually works well as it gives good contrast with the coloured symbols, as does using transparency to show underlying structure.

Here the symbols in the plot above have been changed to rectangular, varying; and display of data values has been turned on.

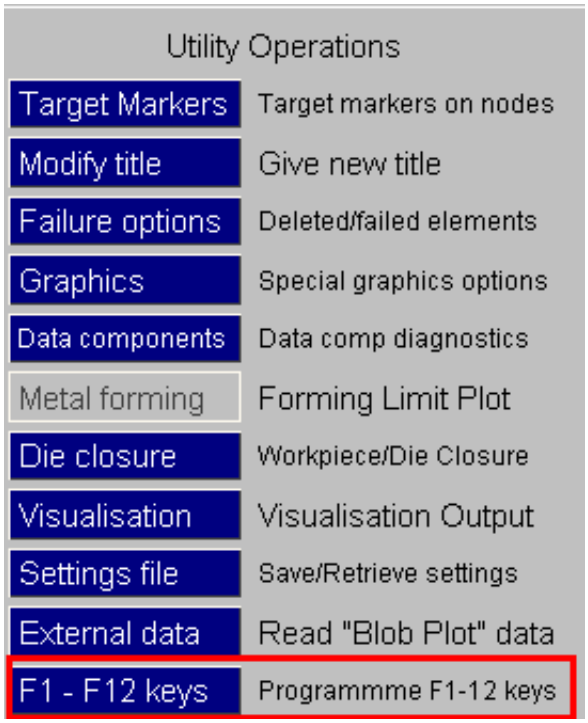


Here symbols have reverted to fixed size squares, and two "beams" have been added to demonstrate their appearance.



Next section

6.9.11 UTILITIES, FUNCTION KEYS



It is possible to programme the function keys F1 .. F12 with command files.

- You create a command file in the normal way.
- The file is associated with a function key.
- It is executed whenever you press that key.

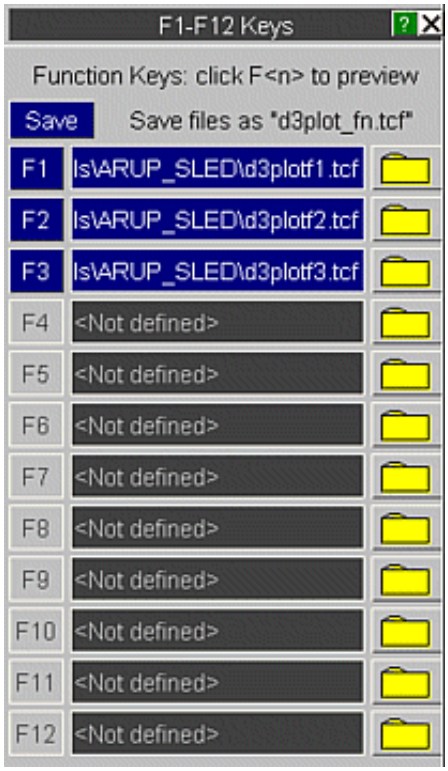
A command file can contain any valid sequence of D3PLOT commands, for example blanking some items and setting a view, or defining a cut-section. Generally it is best to avoid very model-specific commands, or screen picking, if a file is to have a general usefulness on playback. (Command files are described in section 8.)

Files can be associated automatically with function key "***n***" (1 <= ***n*** <= 12) as follows:

- By the "oa_pref" file line **"d3plot*f*n*_key: <***filename***>"**
- By defining file **d3plot_f*n*.tcf** in the current directory (**$cwd)**
- By defining file **d3plot_f*n*.tcf** in your home directory (**$HOME**)
- By defining file **d3plot_f*n*.tcf** in the **$OASYS** directory

Files are searched for in the order given above, and the first definition found "wins". If no definition is found that function key will be inactive.

6.9.11.1 Using the function key panel.

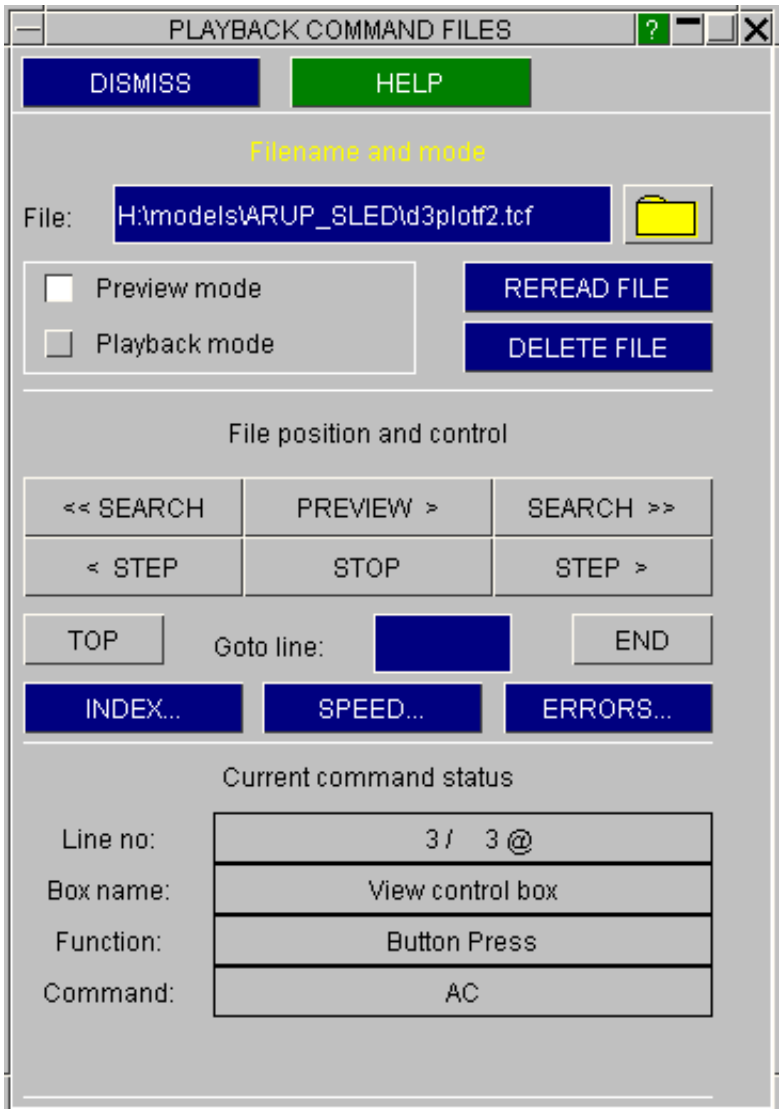


The function key panel shows the current status of all function keys.

You can associate a new command file with a function key at any time by typing in new filename, or use the "?" file filter to find new keys.

SAVE... copies all current definitions to d3plot_f1.tcf ... d3plot_f12.tcf in the current directory. These local copies make it easier to modify standard files for your own use.

Clicking on the **F1** to **F12** buttons will Preview the command file in the standard playback panel. This simply steps through the file's contents without actually executing it:



6.9.11.2 Function key F10 is reserved on Windows platforms.

On Windows platforms the F10 key is reserved by the Windows operating system and cannot be used for D3PLOT macros.

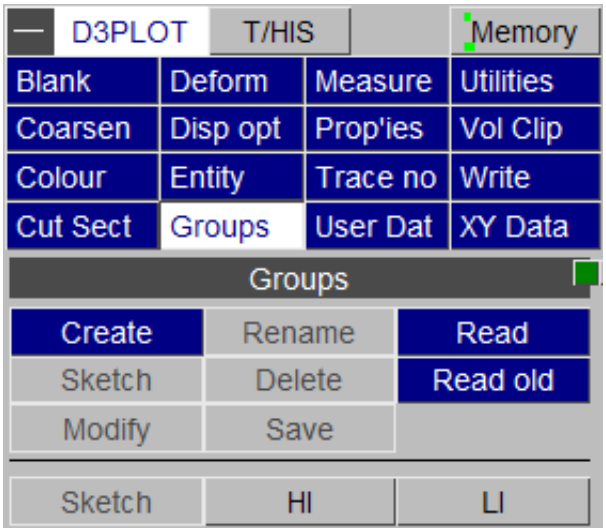
You can still define a function for key F10 in the oa_pref file, so files set up on Unix or Linux machines will still read in, but the key will not work and it will be necessary to associate the macro with a different key.

A warning is issued if you attempt to define a macro for the F10 key on these platforms.

6.9.12 COMPRESS

This option can be used to generate a new set of PTF/D3PLOT files for a model that contain only a subset of the Parts and States in the original model. See section 6.7.7 for more details.

6.10 GROUPS:



Groups as of V9.0 of D3PLOT (Nov 2003) have been extensively rewritten to remove some limitations and to provide new features. In particular:

- You may now have an unlimited number of groups. The previous limit of 30 is removed.
- Groups are now automatically stored in a "**.grp**" file for this model, and are "remembered" across successive runs of D3PLOT, with no intervention required by the user. "**.grp**" files are model dependent.
- The ascii *GROUP format used by Primer may be read in as "**.asc**" files, and converted to internal groups. These files may be created simply by hand, and are model independent. "**.asc**" files may also be written. They are also read by Oasys T/HIS, and this permits groups to be used consistently across the Oasys 9.0 software suite.

Groups may be used in most contexts where selection takes place, for example **BLANK**, **WRITE**, etc.

They can be set up for a model in any of the following ways:

- Defined interactively during a D3PLOT run using **CREATE**.
- Read in from an external ascii (**.asc**) groups file
- Read in from an "old" binary (**.bin**) groups file
- Reread from a previous D3PLOT run, made available automatically via the <*jobname*>**.grp** file.

For backwards compatibility the "old style" binary groups files "**.bin**" may still be read (see section 6.10.8 below), but they are no longer written and their use is deprecated.

CREATE

Create groups by selecting an entity type (e.g. PART), then selecting using the normal methods. You must **STORE** the group to make it available for blanking and other operations. **STORE** saves the group to memory, but not to disk.

SAVE

SAVE stores ***all*** selected groups in an ascii (.asc) groups file, making the groups portable across different models.

The format of this file is described in section 6.10.1 below.

READ... Read groups in from an Ascii groups file.

You can read in groups from any ascii groups file, including one not written from this model. The external data is converted to internal format and saved in the model's **`.grp`** file.

The following rules apply:

Matching of external (file) and internal (model) data

- Matching of data between model and group file is via external labels.
- If an item exists in the model, but is not defined in the file, it will not be grouped.
- If an item exists in the file, but not in the model, it will be ignored - this is not an error.

Groups label and numbering policy:

- Groups in an ascii file have unique labels <#i> <#j> ....
- They will be transferred to become the same internal group labels in the model's **`.grp`** file.
- If group <#n> already exists in the model's **`.grp`** file ***it is overwritten - unconditionally and with no warning.***

Therefore you may read in any number of ascii group files, and the most recent definition of a group in such a file will

be the current one.

Automatic reading of ascii groups files.

Two types of default ascii groups files may optionally be read in automatically when a new model is opened:

• "Master" group file	This may be defined in the "oa_pref" file (see Appendix II) using the **d3plot*master_group_file: <*filename*>** option. This will be read in every time a new model is opened, and the definitions in it will be mapped onto that model's groups, creating or replacing groups as required.
• "Local" group file in current directory	If the "**Read ASC file (Ascii groups file)**" box on the model reading panel (see section 4.1.1) is selected then the most recent file: **d3plot_<*nnn*>.asc** (eg **d3plot_003.asc**) in the directory where the model is found will be read in, replacing or creating groups as above. Note "most recent" may not necessarily be the file with the highest <*nnn*> number - it will depend on the file's creation date.

These two files are read in the order above, thus a definition in a "local" groups file can supersede one in the "master" one.

READ_OLD... Reading "old" (pre version 9.0) binary .bin groups files.

For backwards compatibility the ability to read older **.bin** binary groups files has been preserved, however D3PLOT can no longer write these files.

These files contained up to 30 groups which, in pre 9.0 versions, were hard-wired as labels 1 to 30. When read back in the following rules are applied:

Matching of internal and external data.

- The same rules that are used for ascii groups are applied to binary ones.
- Items in the **.bin** file, but not in the model, are ignored.
- Items in the model, but not in the **.bin** file, are not grouped.

Group labelling.

- Each group in the **.bin** file is stored as the first free group id for this model.
- Therefore assigned group ids will depend on what has been created &/or read in previously.
- You are informed about the label assigned to each group as it is read in.

The use of "old style" binary (.bin) groups files is deprecated.

Binary groups files can be written by earlier versions of D3PLOT, and also by Primer. However their use is discouraged, and the much more compact and easily edited ascii groups files are recommended instead. This is because:

- Ascii groups files tend to be much more compact.
- They are easily created and modified by humans using a normal text editor.
- They can be read by Oasys PRIMER as part of a keyword deck (the special Oasys *GROUP keyword after *END)
- They can be read by Oasys T/HIS.
- And as ASCII format files there are no issues of binary compatibility across different machines.

6.10.1 The format of the ASCII groups file (.asc file)

A file contains one or more ***GROUP** definitions, formatted as follows. All entry is free format.

<table>
<tr><td>*GROUP
<title>
<label></td><td>This is a header designating a new group.
<title> may be up to 80 characters wide
<label> is in the range 1 to 99999999 (However large numbers are deprecated)</td></tr>
<tr><td><item> <range>
<item> <range>
<item> <range>
etc</td><td>
<table>
<tr><td><item> must be one of</td><td colspan="3">PART, SURFACE, NODE, SOLID, SHELL, BEAM, TSHELL, MASS, DISCRETE, SEATBELT, ACCELEROMETER, SLIPRING, PRETENSIONER, JOINT, RIGIDWALL, SEGMENT.

If preceded by a minus sign (eg -PART) then these items are removed from the group.</td></tr>
<tr><td rowspan="3"><range> must be one of:</td><td>ALL</td><td>all items in that category</td><td>eg PART ALL</td></tr>
<tr><td> : <j></td><td><start> to <end> range</td><td>eg SHELL 10 : 200</td></tr>
<tr><td><i j k l m></td><td>Up to 5 discrete labels on a line</td><td>eg SOLID 10 20 33 45 200</td></tr>
</table>
As many lines as required to define the group may be used.
</td></tr>
<tr><td>BOX suffix is Unsupported</td><td>The "proper" group definition in Primer also supports the suffix BOX <label> at the end of a line, meaning that the preceding definitions on that line are limited to what lies within the *BOX definition.

Because D3PLOT doesn’t "know" about boxes these are ignored, and you should avoid the BOX suffix if you are planning to use group files outside Primer.</td></tr>
<tr><td>*END</td><td>This is optional, and is taken to mean the end of the file. A physical <end of file> is treated as terminating the last group in the file, and no *END is required.</td></tr>
</table>

Any number of group definitions may exist in a file, with the next ***GROUP** header effectively terminating the previous group definition.

The easiest way to create a groups file is to write one from D3PLOT, and then look at it in a text editor. The format is extremely simple and easy to understand.

Warnings:

a) D3PLOT groups contact surfaces, if they are present, by interface segment.

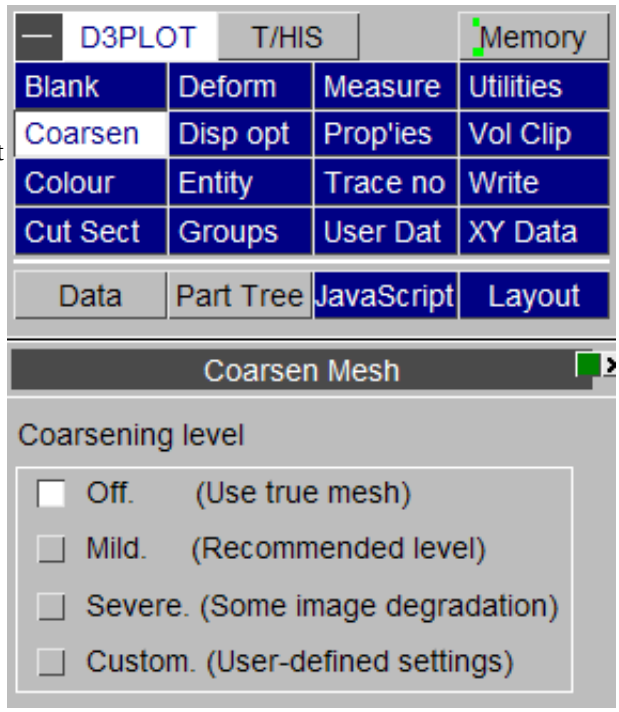
Since segments are not numbered in a LS-DYNA input deck, or indeed may not even be present for contacts defined by anything other than segment sets, attempting to read these back into PRIMER will fail and such definitions should be edited from ascii groups files if they are to be used for this purpose.

b) Group files written by Oasys PRIMER may contain **BOX** arguments, limiting the geometric region in which items are included.

Because D3PLOT doesn’t "know" about boxes it cannot apply these, and they are ignored. Therefore if you are intending to create groups that are portable across programmes it is recommended that you do not use the **BOX** argument.

Next section

6.11 Coarsen



Mesh coarsening is designed to improve the graphics speed of large and finely meshed models. It works by grouping adjacent elements into "patches" and hence sending fewer rendering requests down the graphics pipeline.

It results in a slight loss of image quality, but this is usually acceptable and by default D3PLOT reverts to the original (fine) mesh when you zoom in to look at details.

By default (ie if you don't use "Custom" settings) coarsening does the following:

- Only polygon fill (ie shaded, "fuzzy" shaded image and hidden infill) is coarsened.
- Wireframe and hidden overlay are not coarsened.
- Coarsening is based on the last state in the analysis.
- Zooming in by a factor of more than 4x the autoscaled scale reverts to using the original mesh.

Coarsening is fully automatic, with no user intervention required unless you want to change the default settings.

By default Coarsening is not active, but you can modify this with the **`d3plot*auto_coarsen`** "oa_pref" file option so that large models are automatically coarsened when opened. See Appendix II for more information.

6.11.1 Coarsening Levels, and what they mean.

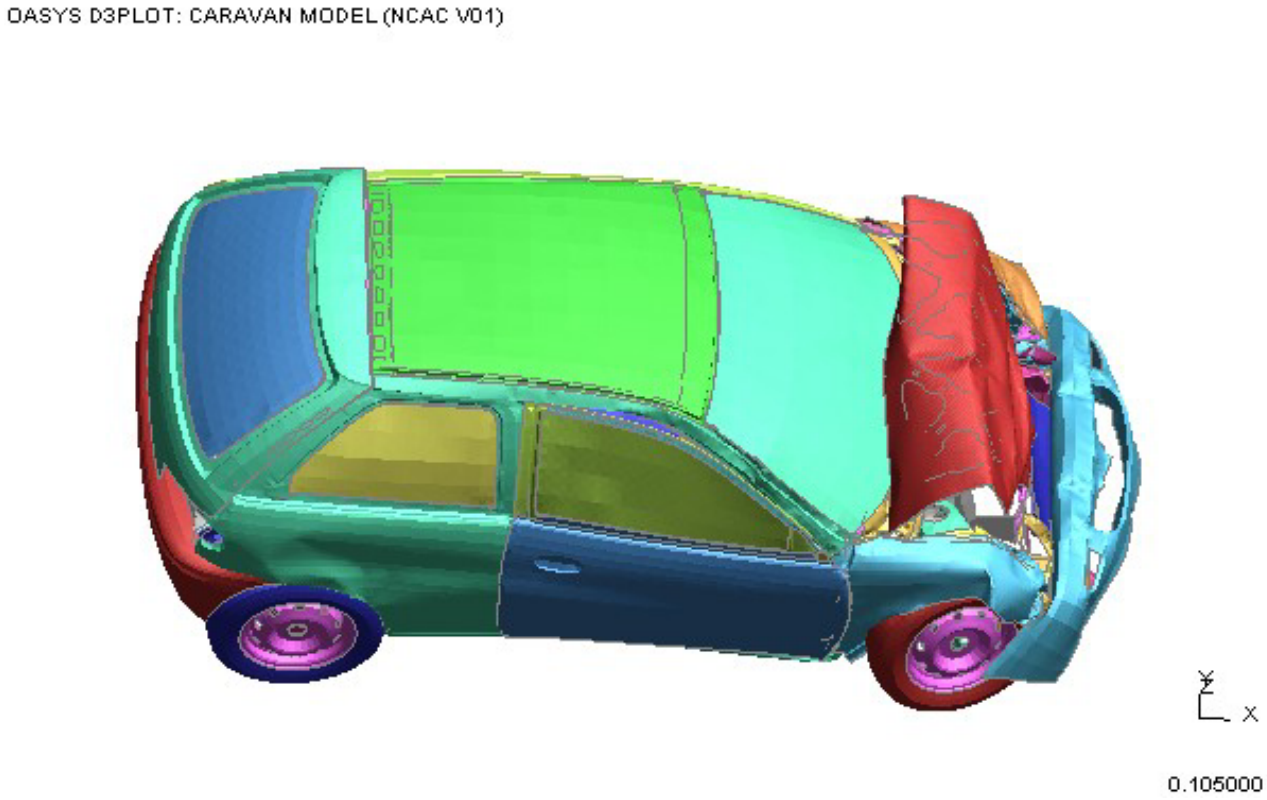
Off	This is the default state. No coarsening takes place and the original mesh is used.
Mild	Coarsening takes place to a reasonable degree, usually with only mild visual artefacts appearing. Typical speed-ups of the order of 40% - 50%
Severe	More aggressive coarsening is used, usually with significant visual artefacts, however larger models can double in speed or better.
Custom	Allows user control over all the coarsening parameters.

Mild coarsening is the recommended level for most models, as it tends to give the best trade-off between speed and image quality.

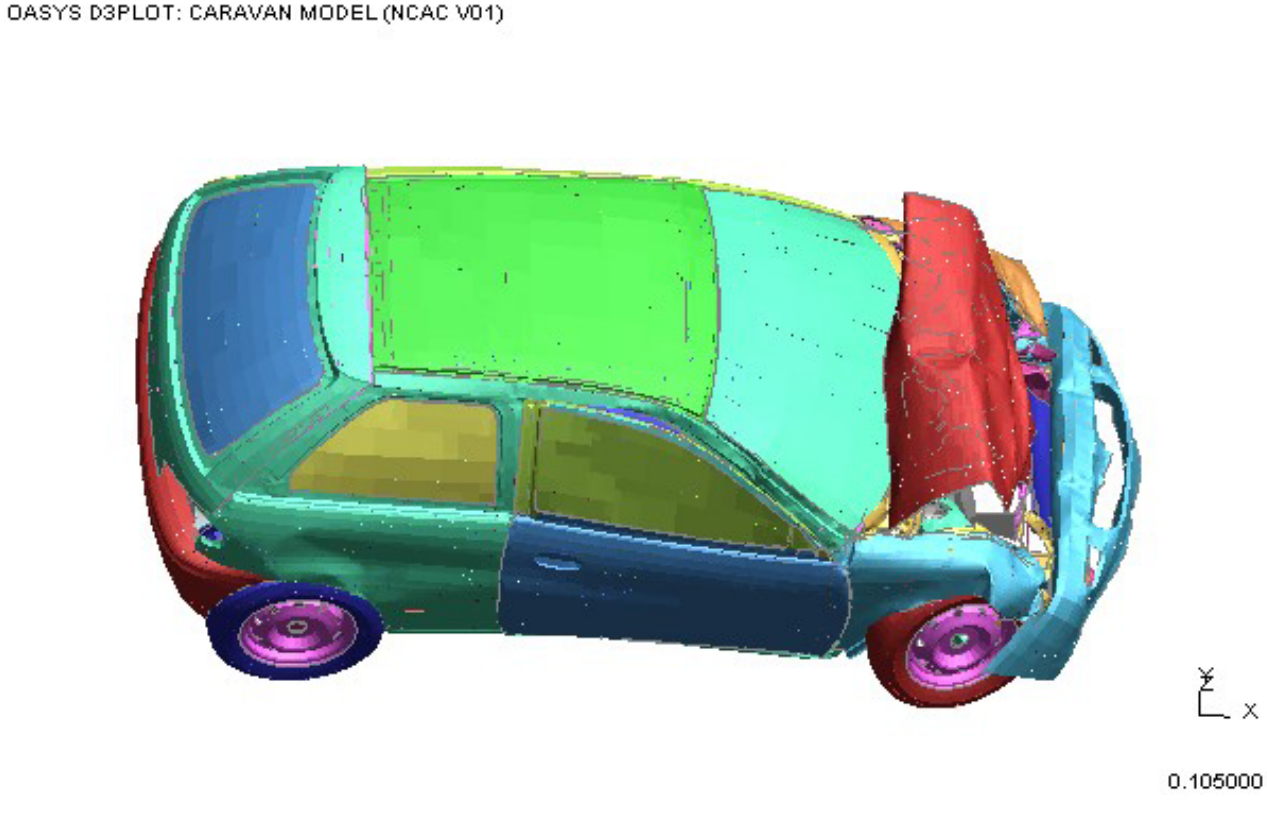
6.11.2 Examples of Coarsening

The images below show a model with approximately 1,000,000 shell elements in its raw state, and illustrate how progressively harsher coarsening reduces the image quality but speeds up the graphics.

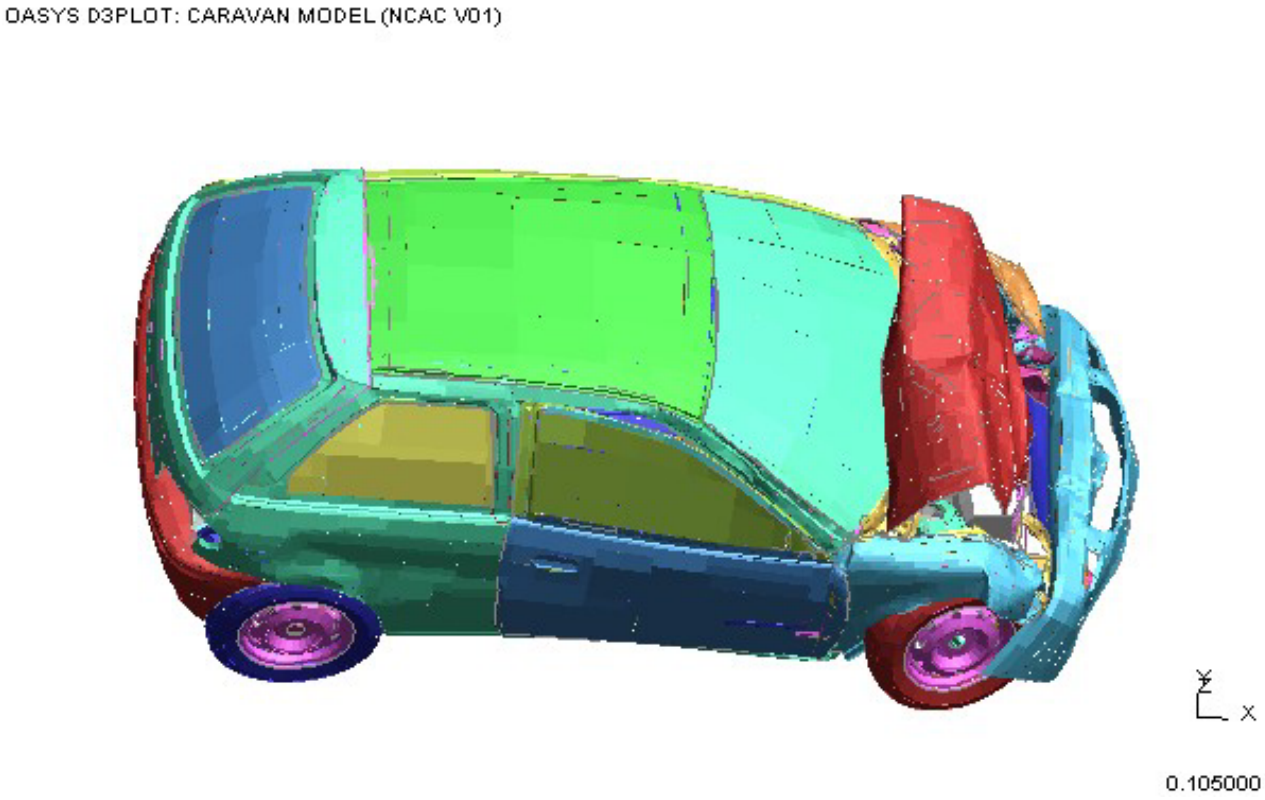
This is the original image. Time to animate a frame on representative hardware: approx **95mS / frame**



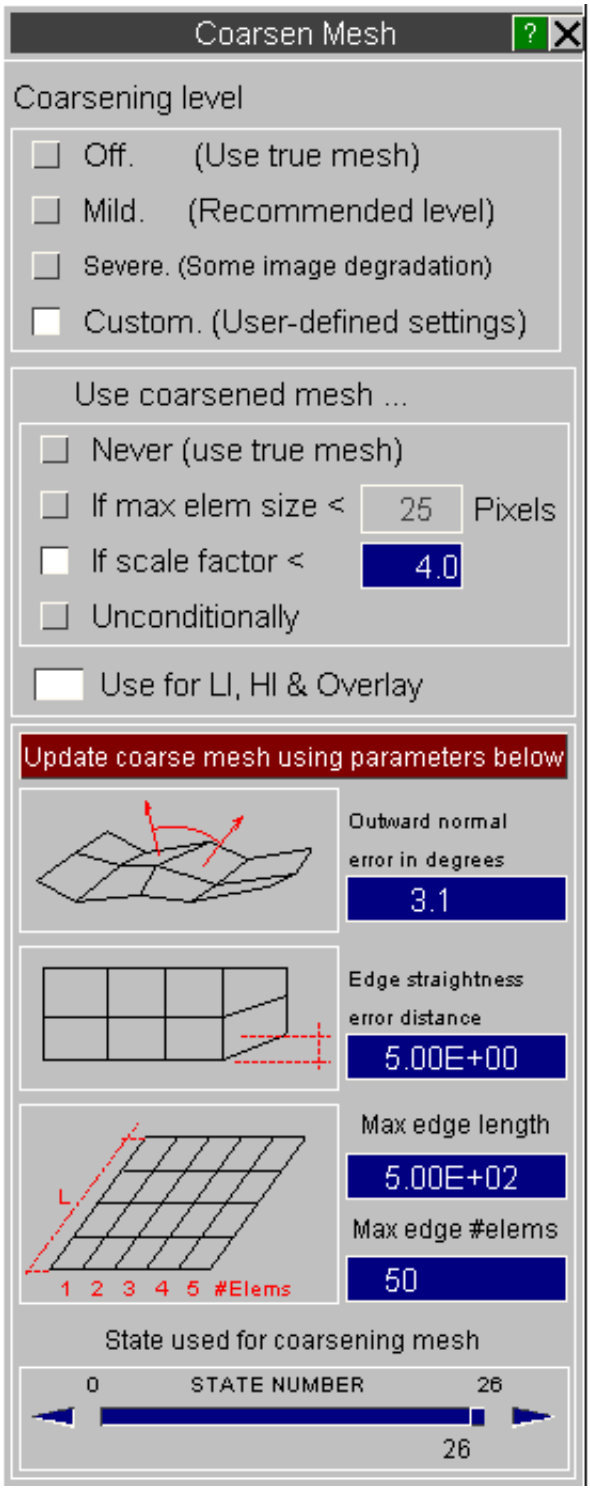
This is the same model using **Mild** coarsening. Some "speckling" is evident and the overlay is poking through in a few places. However time to animate this model is now reduced to about **50mS / frame.**



Finally here is the same model using **Severe** coarsening. More artefacts are visible, and the large patches are clearly evident on the glass panels. However time to animate now approx **30mS / frame**, which is over 3x as fast as the original model above.



6.11.3 **Custom** Coarsening



Use Coarsened Mesh determines the scale at which the transition between true and coarsened mesh takes place.

Never	Means that the true mesh is used at all scales
If max elem size is < xxx pixels	Not recommended, as it requires a knowledge of screen resolution.
If scale factor < fff	This is the factor on the normal "autoscaled" scale for this model. At factors higher than <**fff**>, ie when zoomed in, the original mesh will be used.
Unconditionally	The coarsened mesh is used at all scale factors.

Use for LI, HI and Overlay determines whether or not the coarsened mesh lines are used for wireframe plots, and overlay on shaded and contoured plots.

Normally you should leave this box unchecked, as the coarse overlay is not representative of the true mesh, however it can be useful when tuning some of the other parameters.

Outward normal error, Edge straightness error and **Max edge** attributes.

Play with these at your peril! The default values usually work reasonably well, but they have been arrived at by trial and error over a wide range of models rather than by any scientific means. It is beyond the scope of this manual to explain the theory behind them.

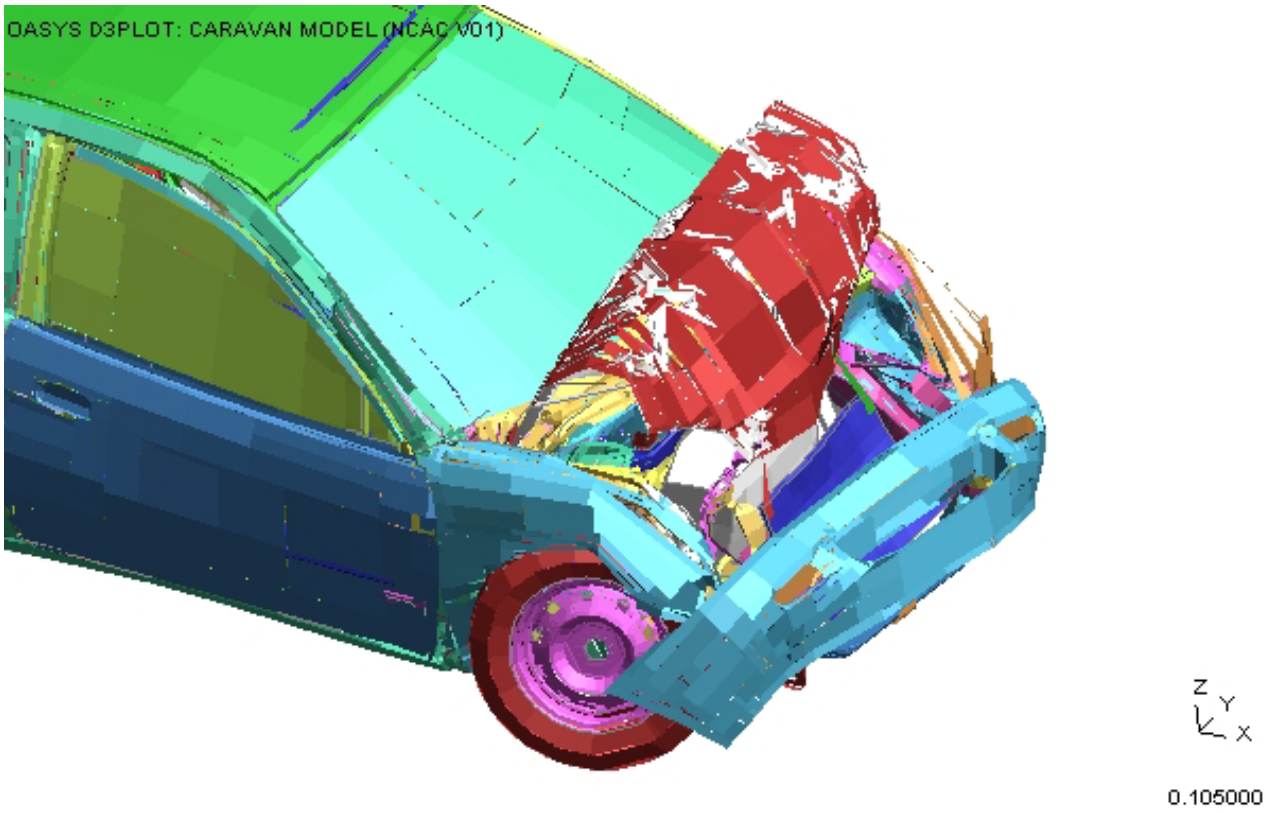
State used for coarsening mesh determines which state is used when extracting the coarse "Patches". An example of using this is given below.

Normally this should be the last state, as this will usually have the greatest deformation and hence limit the elements that can be merged into patches. However if the last state is corrupt, or you are only displaying earlier states, you may get an improvment in both image quality and speed if you choose an earlier state.

Update Coarse mesh using parameters below. If you modify any of the detailed parameters the coarse mesh will not be recalculated, and the changes will not therefore take effect, until you press this button.

What happens when you base coarsening on an earlier state.

In this image, using the same model as above, **Severe** coarsening has been used but has been based on the undeformed geometry at state #1. The errors that arise from basing patches on elements which subsequently become deformed are obvious!



6.12 **T/HIS** the D3PLOT <=> T/HIS link

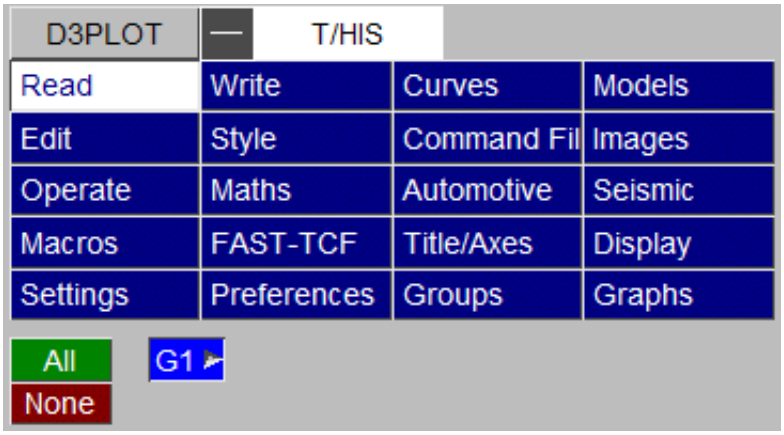
The **T/HIS** command starts T/HIS in background and establishes a link between the two codes, allowing D3PLOT to access and display "time-history" data, and to synchronise time-history display in T/HIS with the current state displayed in D3PLOT.

Linked T/HIS has the following attributes

- One or more fully functional T/HIS graphics windows running within the D3PLOT user interface.
- The two codes "talk" to one another and share information about time-history data.
- Time-history "timeline" plots can be synchronised to show the current state in D3PLOT
- Accessing a particular time in either code will cause the other to update to show the same time
- Plotable items in time-history plots are displayed in D3PLOT, and may be screen-picked for selection in T/HIS.

When the T/HIS link is active the user interface operates either in D3PLOT native mode (as in the rest of this manual), or T/HIS native mode. You need to be in the correct mode to access commands for the relevant program, and you swap between modes using the **D3PLOT** and **T/HIS** buttons on the top right.

6.12.1 The **T/HIS** panel

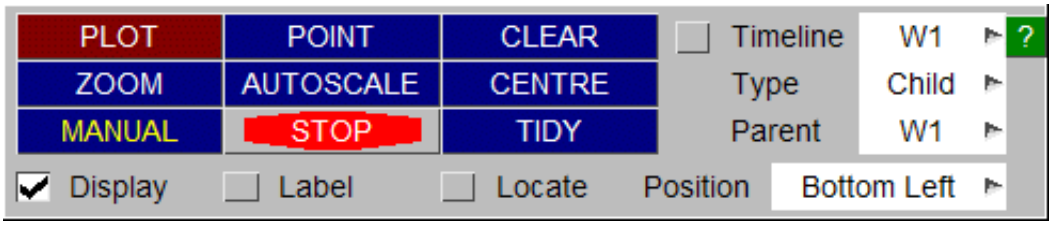


When you first click on **T/HIS** the following happens:

- D3PLOT silently opens a background T/HIS process
- This is passed the name of the current model.
- It searches for files called <**name**> that are readable by T/HIS.
- If found these are scanned and read into T/HIS

As shown here the top level T/HIS panel is identical to the "native" T/HIS top level commands menu, with the addition of some options specific to the linked case.

In addition to the top level panel changing to display the T/HIS commands the D3PLOT viewing menu is replaced with the T/HIS Global Command menu.



If the top level panel is changed back to display the D3PLOT commands the D3PLOT viewing panel will be redisplayed.

6.12.1.1 Association between models in D3PLOT and T/HIS

When T/HIS is started it will scan for time-history information for all models currently open in D3PLOT, and will continue to maintain associativity between the two codes. For example model M2 in D3PLOT will also be model M2 in T/HIS.

As models are opened and closed in D3PLOT so T/HIS will also open and close them, maintaining parity of model numbering. Internal flagging within D3PLOT of items being available for time-history processing will also be maintained.

WARNING: Opening a model in linked T/HIS only, rather than via D3PLOT, breaks the associativity between the two codes.

It is not illegal to open a model directly in linked T/HIS, but it has side-effects:

- Because D3PLOT cannot "know" about such a model's attributes it cannot be processed in D3PLOT in any way
- To make this clear models opened in this way are given model numbers 101, 102, ... in T/HIS

6.12.1.2 Use of T/HIS derived time-history data in D3PLOT

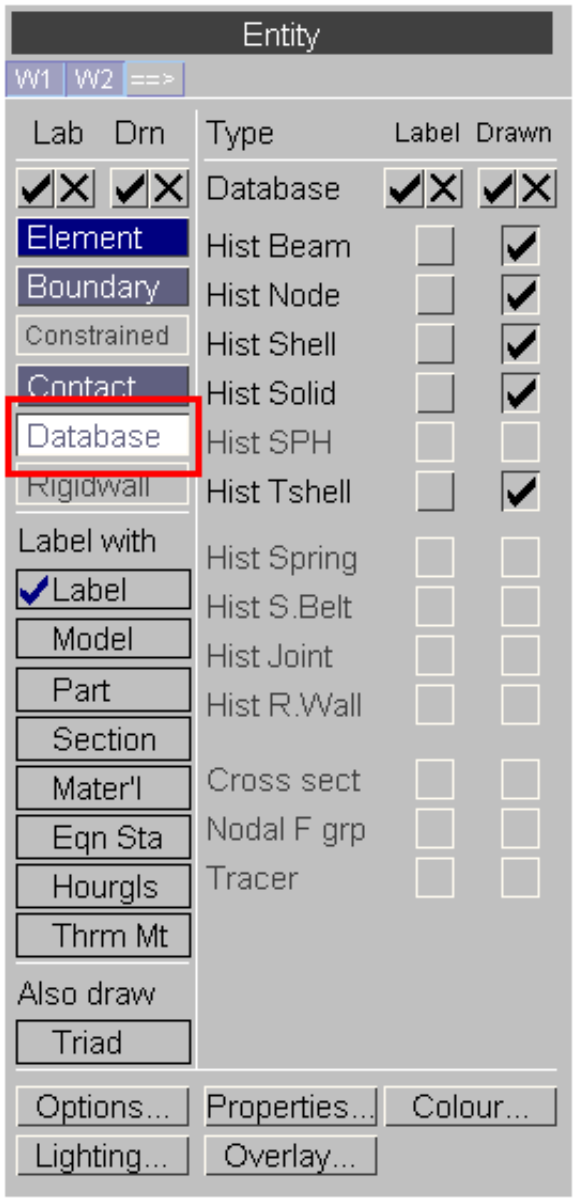
D3PLOT can display several items (for example springs, joints, elements and nodes in time-history blocks) that can have "time-history" information associated with them. When the T/HIS link is active this is handled as follows:

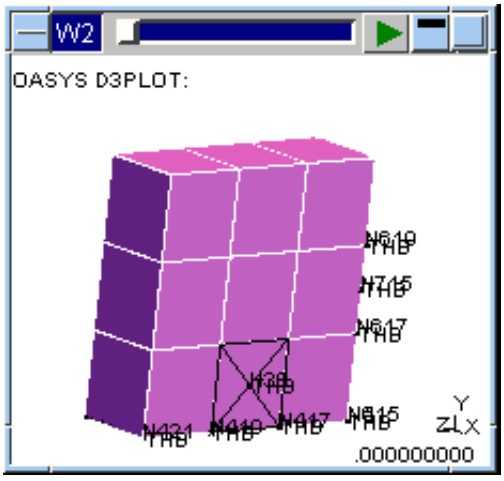
- Items in time-history blocks (ie those with **`*DATABASE_HISTORY_xxx`** cards) may be displayed by using the **Entity, Database** panel as shown here.

  In this example there are beams, nodes, shells, solids and thick shells in time-history blocks; but while two models are current only one of them contains time-history data, which is why the tick boxes have a grey background.

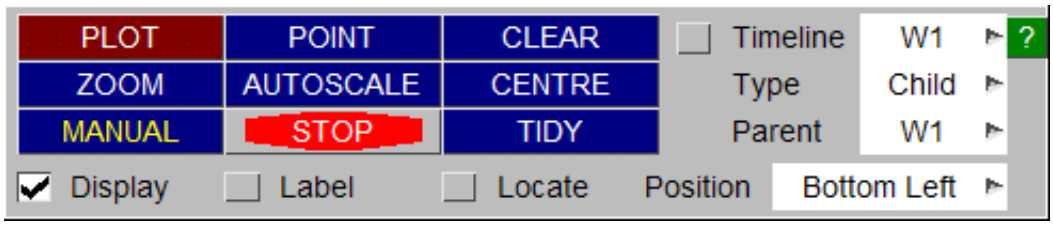
- Time history block symbols are displayed on elements as crossed lines, and on nodes as small squares. The image below shows the display of a solid and a few nodes in time history blocks (with labels turned on).

- Items not explicitly on *database_history cards, but which have time-history data available (springs, seatbelt elements, joints, rigid walls) may also have their visibility switched in this panel as "pseudo database history" items.





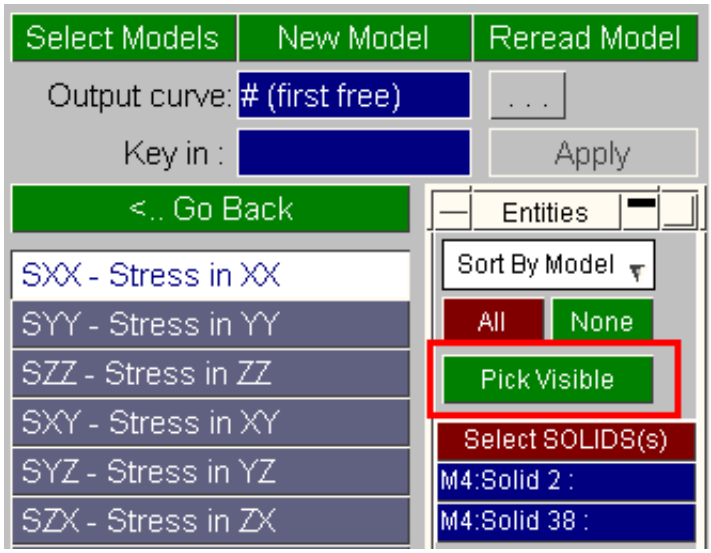
The display and labeling of items in "time-history" blocks can also be controlled using the **Mark Items** and **Label Items** buttons in the T/HIS Global Command menu.



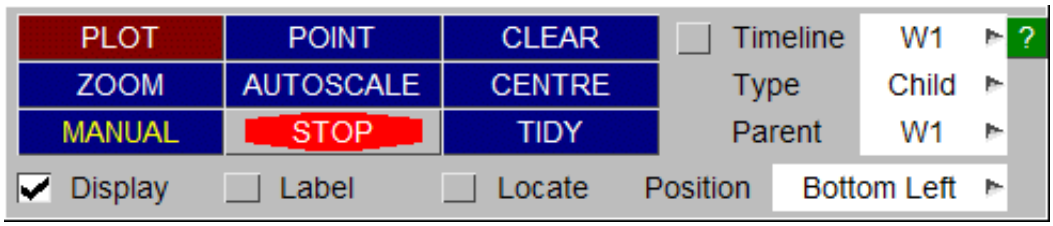
Screen-picking items in Time-history blocks for selection in T/HIS

When items in time-history blocks are displayed in D3PLOT they can be screen-picked for selection in T/HIS menus by using **Pick Visible**.

Screen picking, menu selection and **Key in** can all be combined at will for the purposes of item selection.



6.12.2 Linked **T/HIS** commands

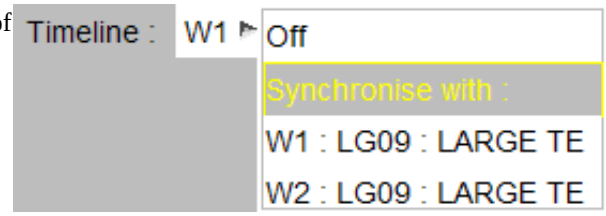


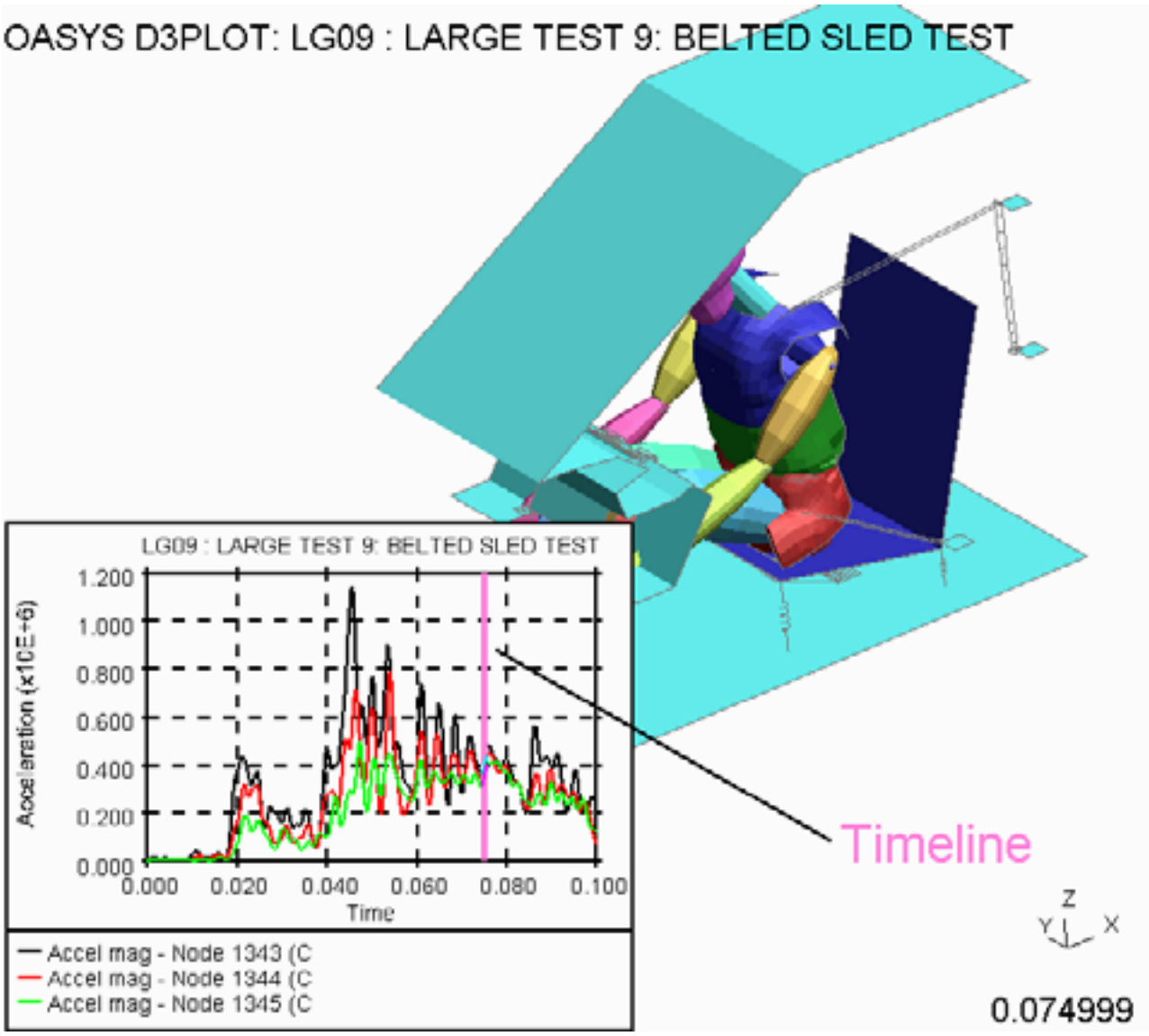
6.12.2.1 **TIMELINE**

TIMELINE adds a vertical bar to the T/HIS plot showing the current time displayed in the D3PLOT graphics window. Each T/HIS graph can have it's own timeline.

By default the first D3PLOT window will control the position of the timeline in each T/HIS graph.

The D3PLOT window used to control the timeline position can be changed using the popup menu in the T/HIS Global Command menu. This popup can also be used to turn **OFF** the timeline. This popup menu will set the timeline for ALL the T/HIS graphs.



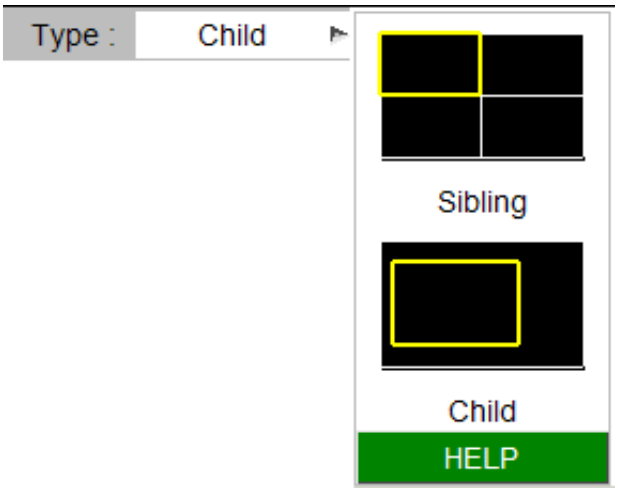


After turning on the timeline in a T/HIS graph

- You can drag this bar in the T/HIS window to a new point, and the D3PLOT graphics window will jump to the new time.
- Equally if you move the D3PLOT graphics window to a new state the bar in the T/HIS plot will update.

Note that the timeline bar can only be positioned at times in the complete state (.ptf) file, since only these times are available as plotting states for D3PLOT. (Although D3PLOT can interpolate between states it would be potentially misleading to associate a "real" time-history value at some intermediate time with an interpolated graphical state.)

6.12.2.2 Graph Type



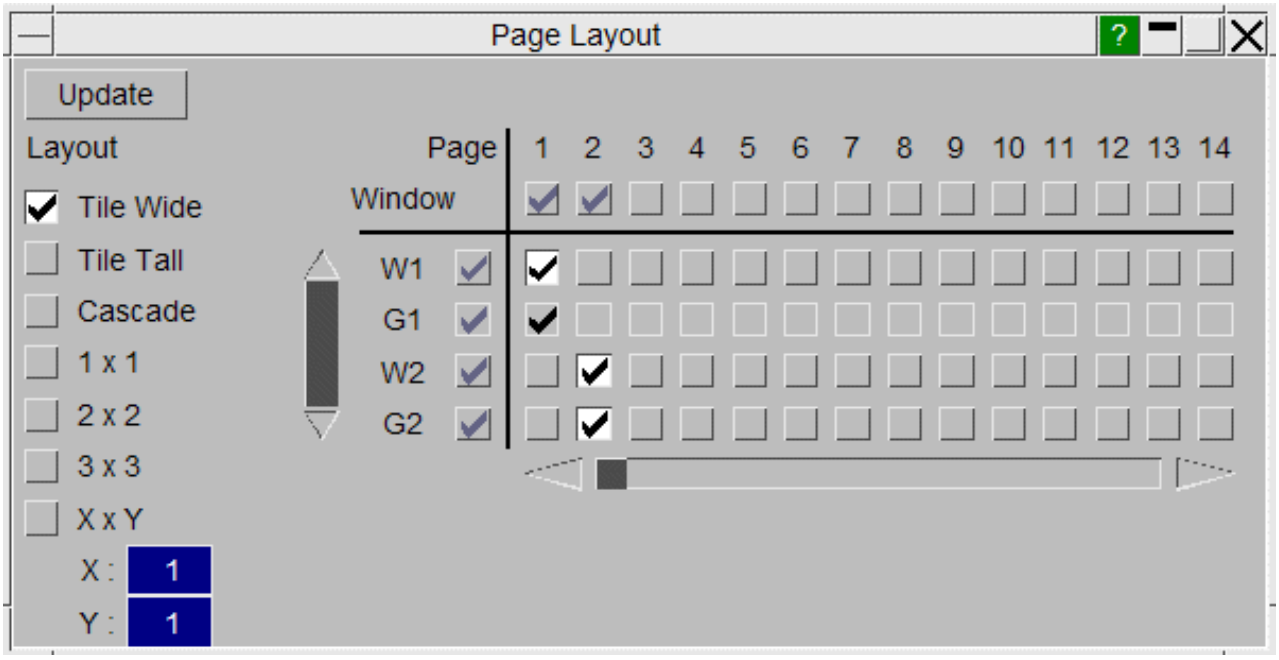
The T/HIS graph windows can co-exists with other D3PLOT graphics windows or they can be located within D3PLOT windows.

The type of each T/HIS window can be set to either

SIBLING	The T/HIS window occupies a "slot" in the graphics window layout, just like the other D3PLOT windows **W1 .. Wn**
CHILD	Makes it a child of a graphics window. If there is more than one you must choose which is to be its parent. It can be resized by dragging its borders, but may also have a preset **Window Position** in one of the four quadrants.

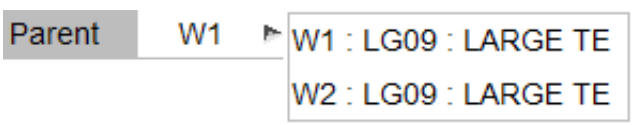
SIBLING T/HIS graphs are positioned on Pages using the same rules as other D3PLOT graphics windows (see Section 2.6.2.1). **CHILD** T/HIS graphs do not take up a slot on pages.

The example opposite shows 2 D3PLOT windows (**W1** and **W2**) and 2 T/HIS graphs (**G1** and **G2**). Page 1 contains **W1** and **G1** (**G1** is a child of **W1**) and Page 2 contains **W2** and **G2**.



CHILD T/HIS graphs can have their window borders removed by docking the windows using the **Dock** option in the Toolbar of each window.

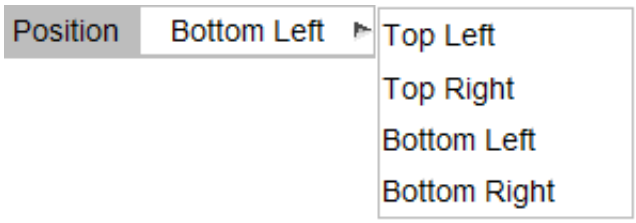
6.12.2.3 Parent



This option only applies to **CHILD** windows.

Each T/HIS graph can be positioned within any D3PLOT window.

6.12.2.4 **Position**

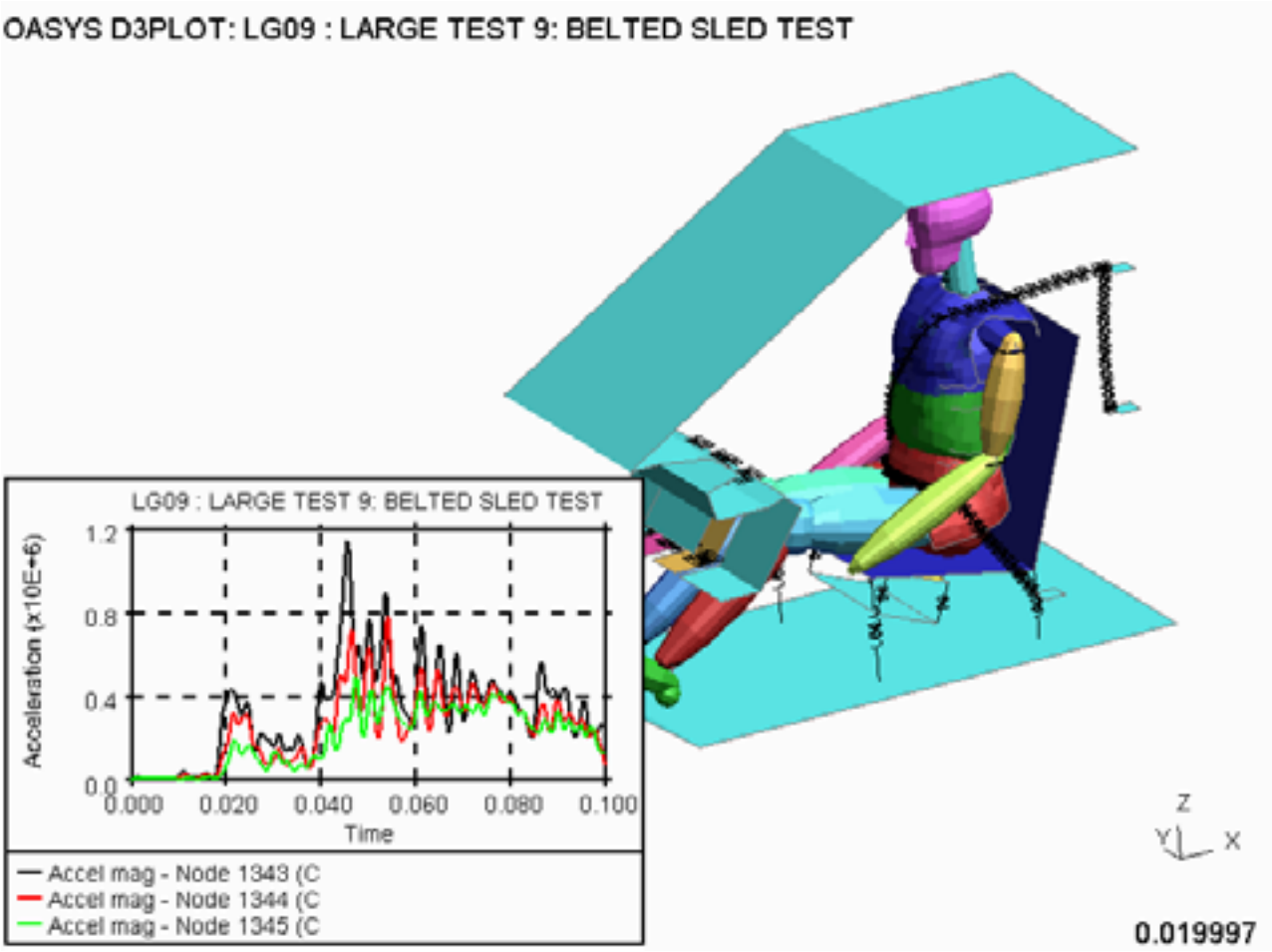


This option only applies to **CHILD** windows.

Each T/HIS graph can be positioned automatically into one of the four quadrants: bottom/top left/right.

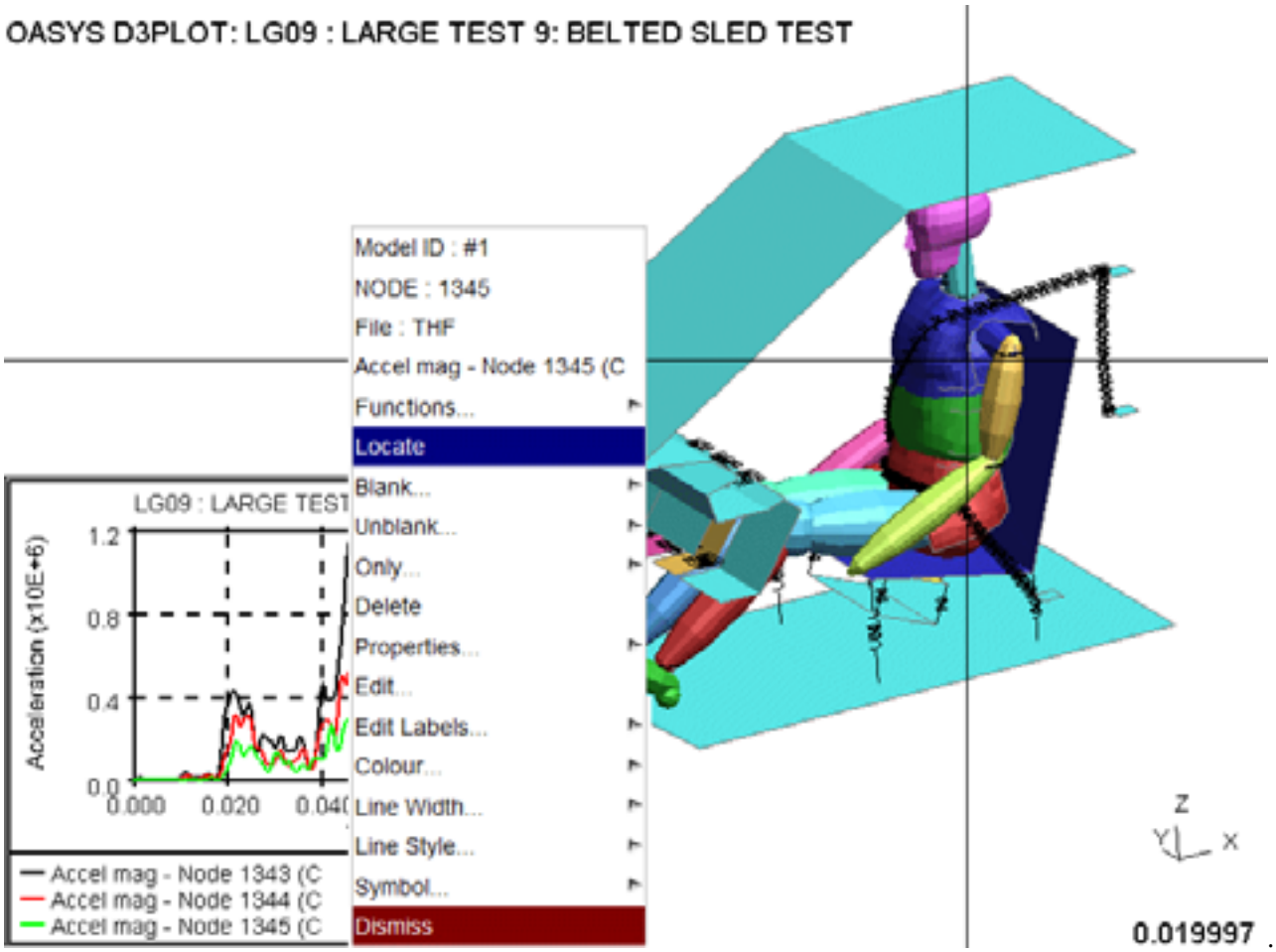
Alternatively each T/HIS window can be moved and resized by clicking and dragging on it's borders and positioned anywhere within the parent D3PLOT window.

This example shows a **DOCKED** window located in the bottom right hand quadrant of the graphics window. No button bar is drawn because the mouse is not in that window.



6.12.2.5 **LOCATE** Identifying a curve's item visually in D3PLOT

The **LOCATE** function lets you right-click on a curve in the T/HIS window to identify the item associated with it. D3PLOT will then draw cross-hairs through that item in the graphics window. In this example node 1345 has been identified



Only one item can be located at a time: each new pick supersedes the previous one. To turn off the crosshairs deselect **LOCATE** in the T/HIS panel.

6.12.3 Setting Properties for Individual Graphs

The options in the T/HIS Global Command menu always apply to all of the T/HIS graphs that are currently active.

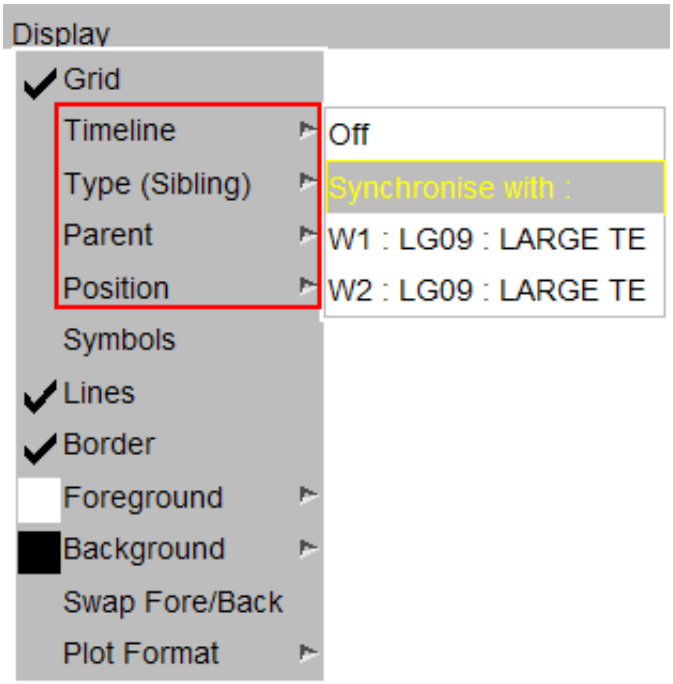
Properties for individual graphs can be set using the **Display** drop down menu at the top of each T/HIS graph.

Timeline This option can be used to turn on and off the display of the Timeline and the D3PLOT window controlling the timeline for a single T/HIS graph.

Type This option control the position of the T/HIS graph in relation to other windows (see Section 6.12.2.2 for more details)

Parent Set the parent for CHILD windows (see Section 6.12.2.3 for more details)

Position Set the position for CHILD windows (see Section 6.12.2.4 for more details)

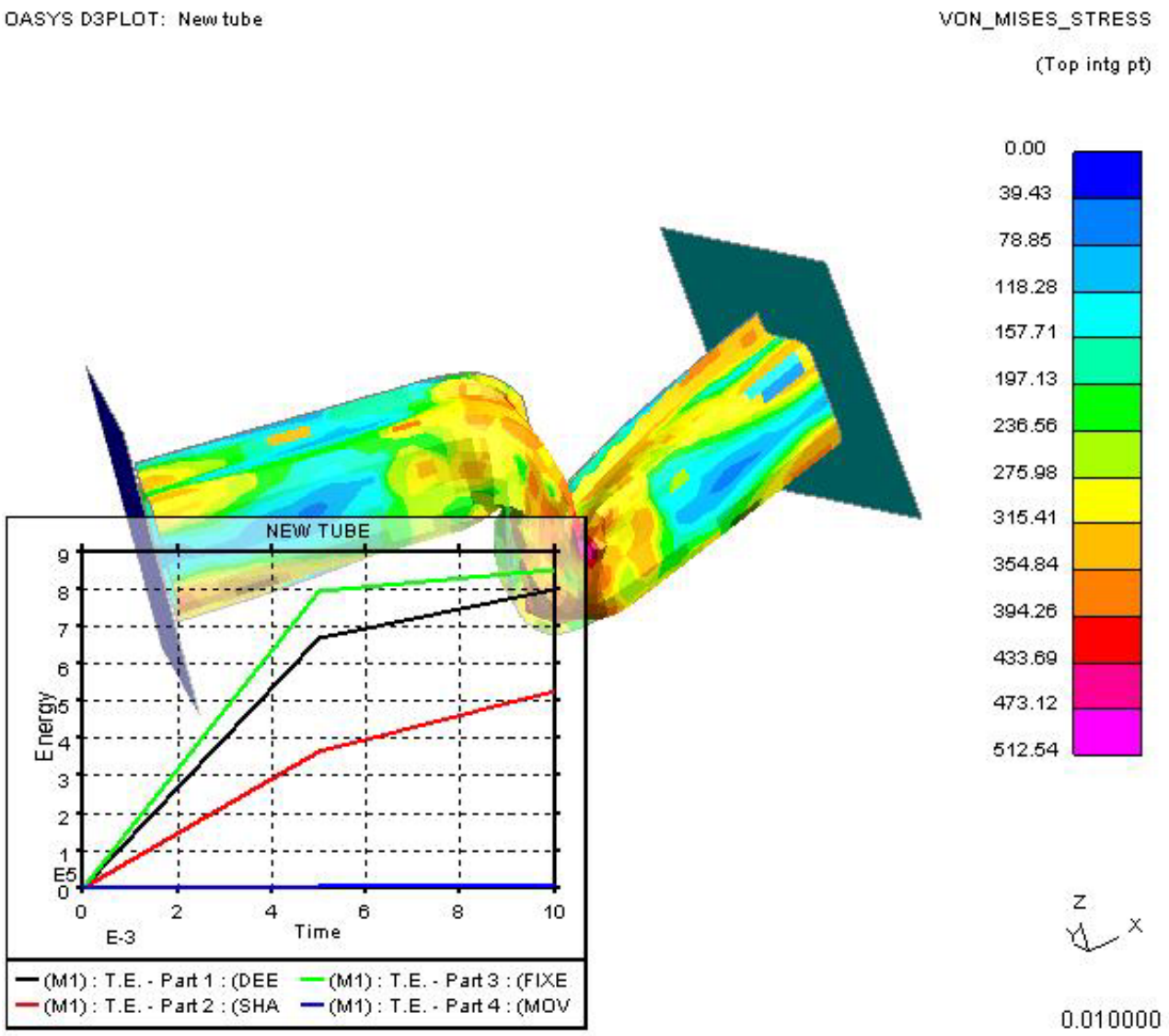


6.12.4 Using **IMAGES** to capture linked images to file.

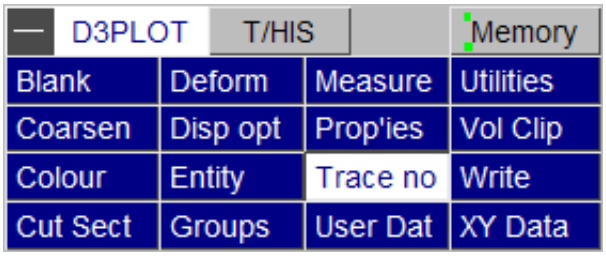
The **IMAGES** command in D3PLOT is capable of capturing images containing both D3PLOT and T/HIS windows, (see Section 7.0 for more details.)

As T/HIS windows can be located within D3PLOT windows the transparency of T/HIS windows can be adjusted so that the underlying D3PLOT image can be seen through the T/HIS window.

The figure below shows an example of a 50% transparent "docked" T/HIS image overlying the D3PLOT one. In the undocked or "sibling" cases a composite image will be the size of the rectangular bounding box required to enclose the selected windows.


OASYS D3PLOT: New tube
VON_MISES_STRESS
(Top intg pt)
0.00
39.43
78.85
118.28
157.71
197.13
236.56
275.98
315.41
354.84
394.26
433.69
473.12
512.54
NEW TUBE
Energy
E5
E-3
Time
(M1) : T.E. - Part 1 : (DEE
(M1) : T.E. - Part 2 : (SHA
(M1) : T.E. - Part 3 : (FIXE
(M1) : T.E. - Part 4 : (MOV
Z
X
0.010000

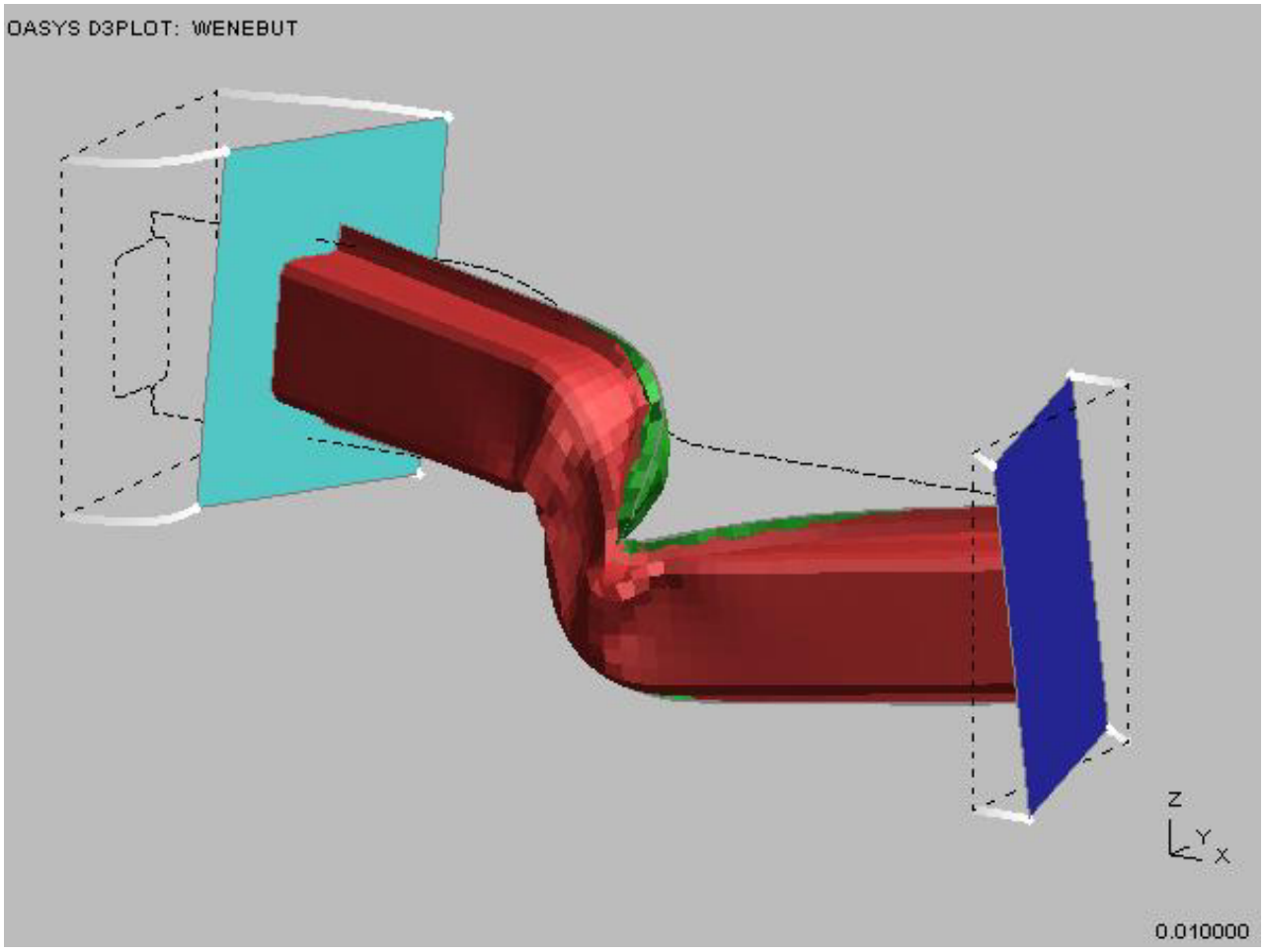
6.13 Trace Node



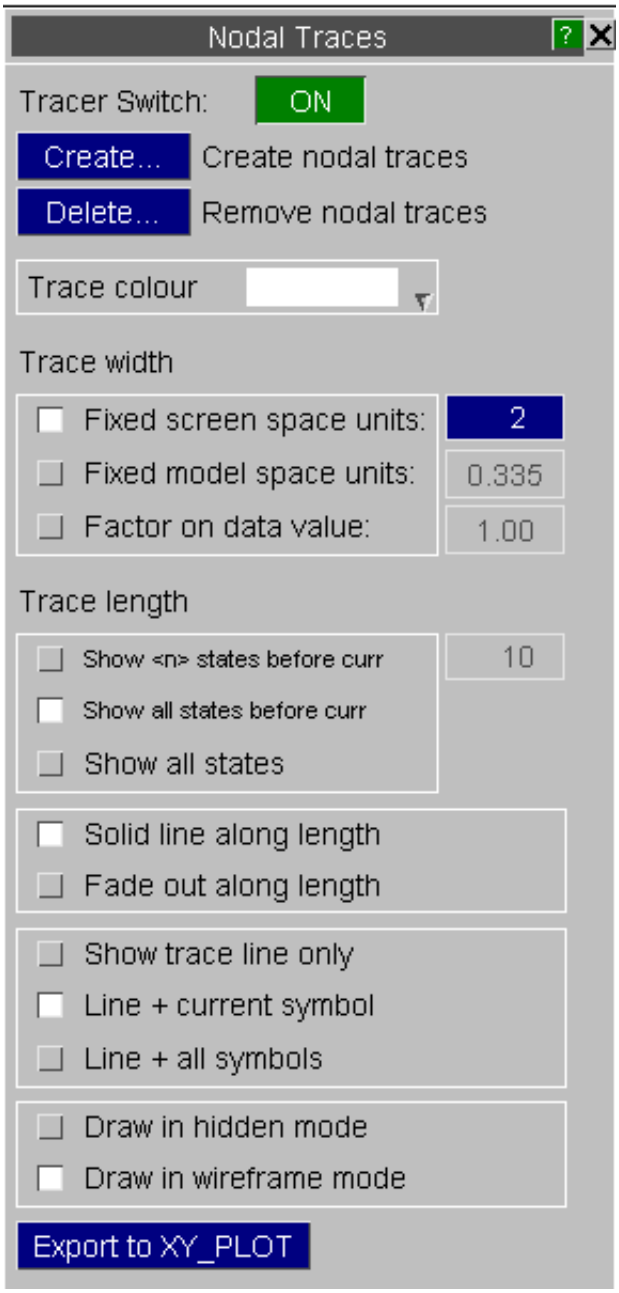
Trace node adds lines ("traces") showing the motion history of selected nodes.

The nodes displayed are user-selected, and the attributes of the trace lines are also selectable.

Here is an example of a crush tube on which the corner nodes of the plattens have been selected for traces. The outline undeformed geometry is also displayed to show how the trace paths display the path from original to current geometry.



6.13.1 The Trace Node control panel



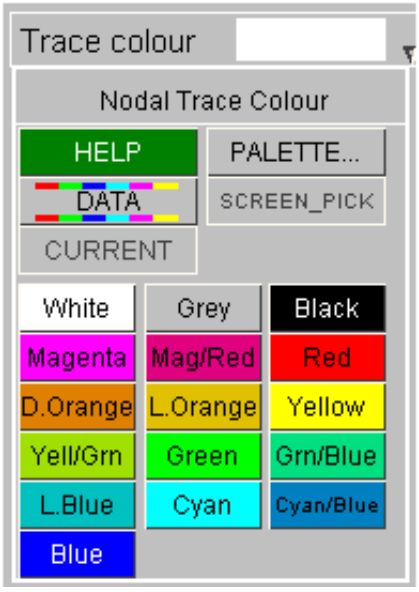
The steps to be gone through are:

1. Use **Create...** to define the nodes for which you want traces. These can be typed in by label or screen picked.
2. Turn the tracer switch **ON** and traces will be added to your plots.
3. You can add more nodes at any time using **Create...** , and likewise delete nodes using **Delete...**

The meaning of the remaining controls is as follows. Click on an item to go to its detailed description.

Trace Colour	Sets the colour to be used. May be fixed or based on data values at the node.
Trace width	Trace line width. May be fixed or based on dat avalues
Trace length	How many points prior to the current node position are shown.
Solid / Faded line	The line can be solid along its length, or can fade out.
Line and symbol display	The line itself is always shown, but you can also choose to show the current and previous symbols
Hidden / wireframe mode	In hidden mode the lines are subject to hidden surface removal, ie they co-exist with other items. In wireframe mode they are effectively drawn on top of any structure and will always be visible.
Export to XY_PLOT	Make a graph of the global co-ordinates of the nodes at each time-state.

6.13.2 Trace Colour



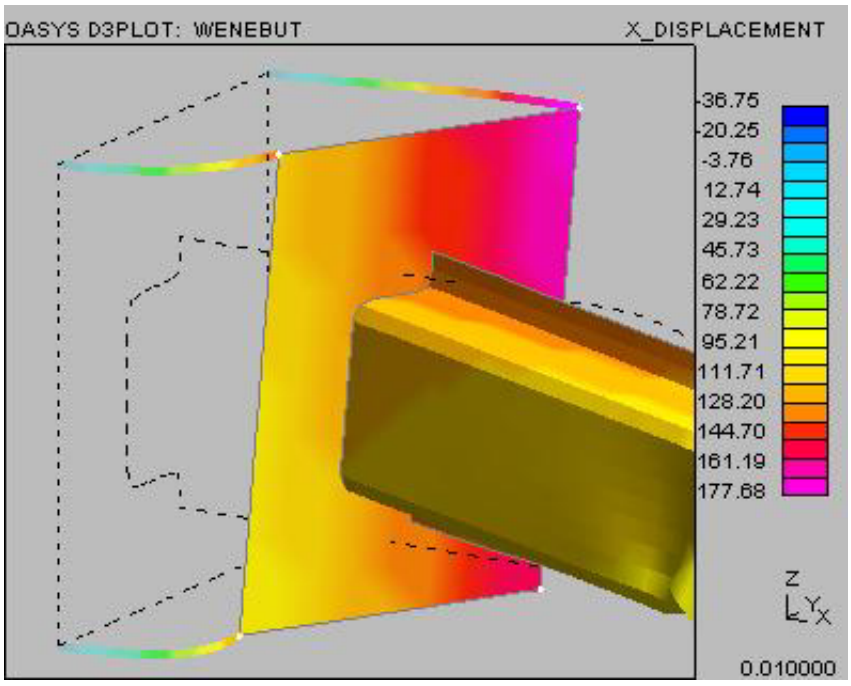
Colours may be constant, chosen from the standard list or mixed using the Palette, this is straightforward.

However

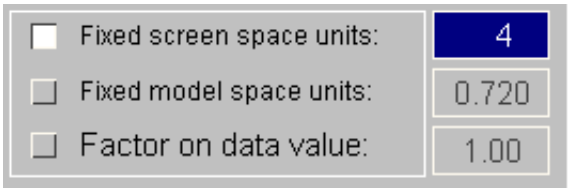
requires some explanation:

If you have previously performed some sort of data-bearing plot that has generated values at nodes then **DATA** colours will draw traces bearing those colours at the selected nodes. This allows you to show how values have changed over time.

For example the image below uses **DATA** mode to display colours showing X displacement on the example above. Here the traces have been added to a shaded image contour plot, but they would still be coloured if added to a non-data-bearing plot.



6.13.3 Trace Width

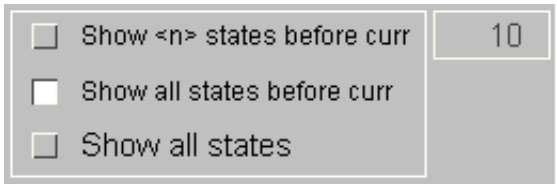


Sets the width of trace lines.

NOTE: Trace widths are clamped to lie between 1 and 25 screen units to prevent plots getting ridiculous in either of the last two cases

Fixed screen space units	Will give a fixed number of pixels width that will not change as the scale of the plot changes.
Fixed model space units	Will also give a fixed width, but in the actual units of the model, so this ***will*** change as the image scale changes.
Factor on data value	Will multiply the data value at the node to produce a quantity that is treated as a screen space unit, so it will not change with image scale

6.13.4 Trace Length



Specifies how many points before the current one are shown in a trace line.

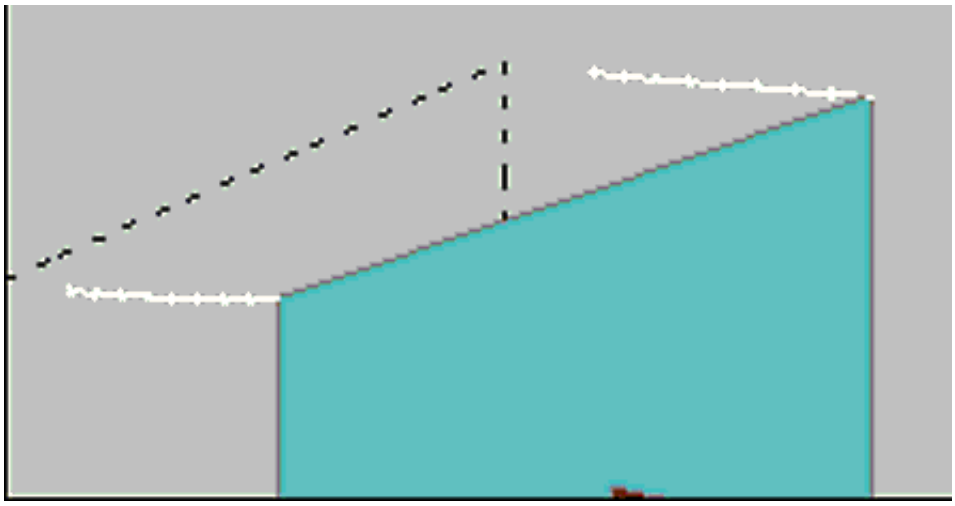
This is best demonstrated by example. In all cases point symbols have been turned on to make clear what is being displayed.

Show <n> states before curr

(Here <n> = 8)

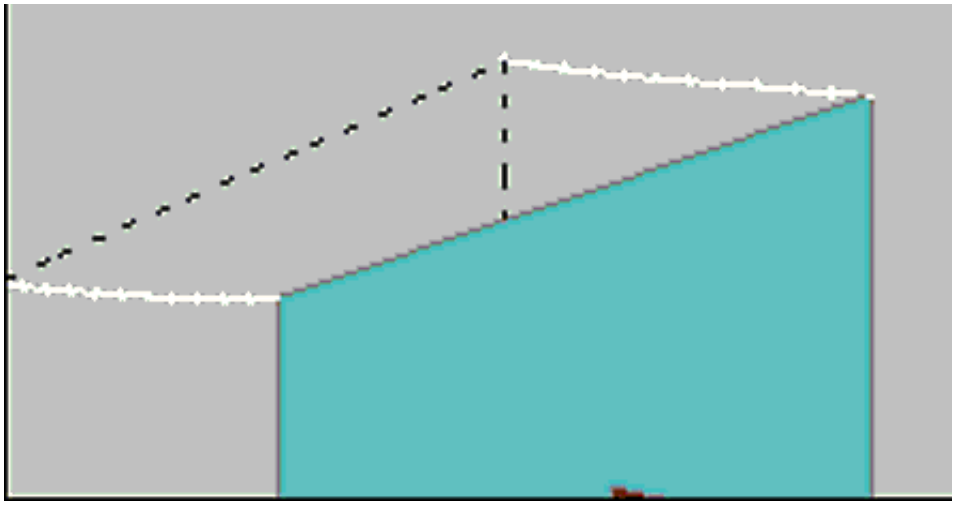
This shows the 8 states prior to the previous one.

If fewer than <n> states are available then only those between the current and state #0 will be shown.



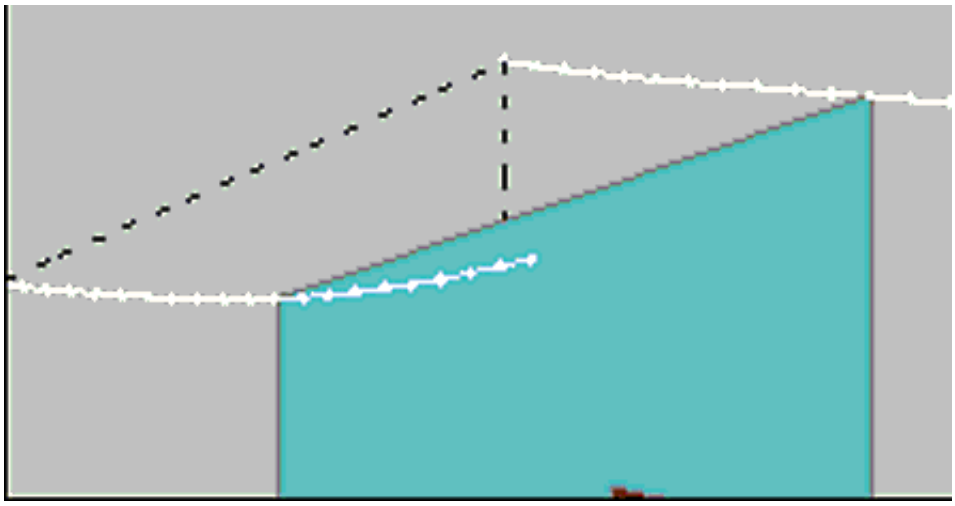
Show all states before curr.

This shows all states from #0 to the current one.

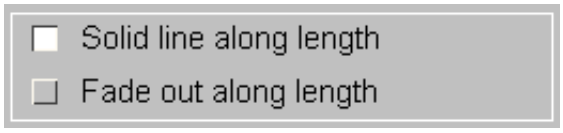


Show all states

This shows all states, including those both before and after the current one.



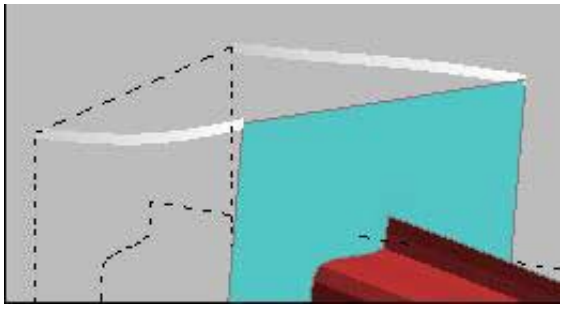
6.13.5 Solid / Faded out line



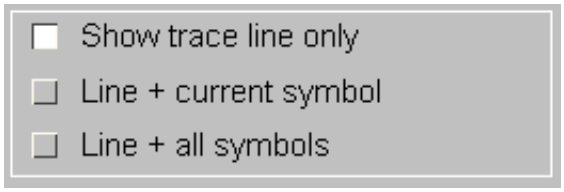
Determines whether lines are drawn solid, or whether they

Solid lines are obvious (all three plots above use them).

Here is an example of a plot fading out along its length, showing that it gives a good visual indication of the "age" of each point compared to the current one.



6.13.6 Symbol Display



Determines whether symbols are shown, and at which positions.

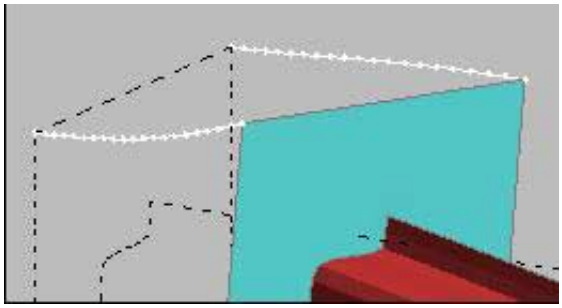
"Symbols" are a diamond shape on the line at a data point.

Show trace line only: No symbols are drawn, and only the line is shown.

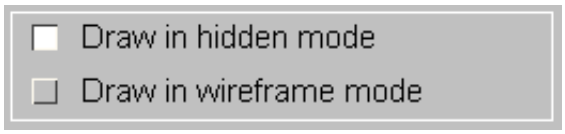
Line + current symbol: The line is drawn, and a symbol is shown at the current point only.

Line + all symbols

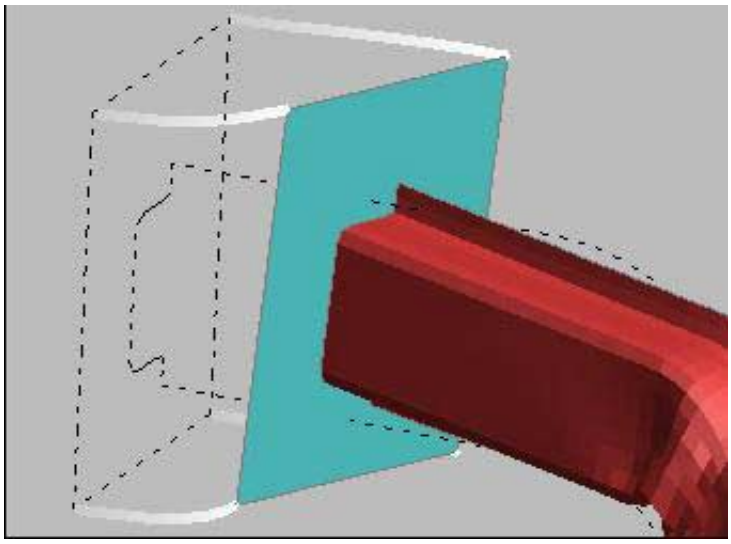
Symbols are drawn at all points, as shown here.



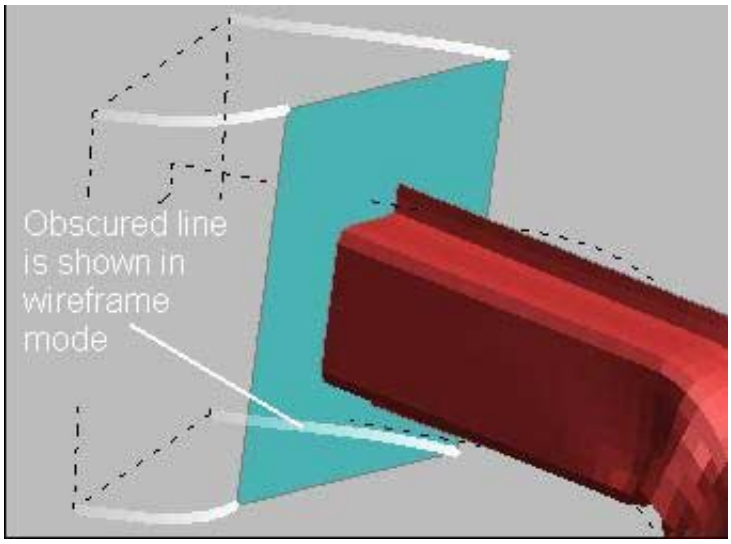
6.13.7 Hidden vs Wireframe mode



In "Hidden" mode trace lines occupy the same 3D space as the model, and may become obscured by structure.



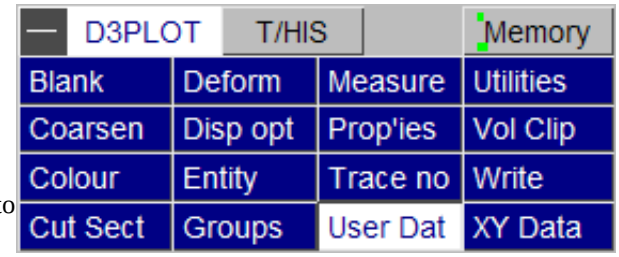
In "Wireframe" mode trace lines are effectively drawn above the model and will always be visible.



6.13.8 XY Plot

Make a graph of the global co-ordinates of the nodes at each time-state. The plots can be saved as curve files or transferred to T/HIS if the T/HIS link is open.

6.14 User Data



"User Data" creates and manages user-defined data components.

These allow completely arbitrary data for nodes and elements to be processed. The data may be generated externally, or computed locally from existing data already present in the database.

Special "binary" user-defined data components may be created and manipulated via the Javascript interface.

6.14.0 Description of User-defined Data components

Conventional data components in D3PLOT are typically nodal displacements, velocities and accelerations, and element stresses, but are limited to what is present in the database that has been read.

User-defined components take this a stage further and allow you to define any number of new data components of the following six types:

Entity type	**Class of data**
Nodes	**Scalar**: A single value at nodes
	Vector: A [x,y,z] data vector at nodes
Solids, shells, thick shells	**Scalar**: A single value at elements
	Tensor: A [xx,yy,zz,xy,yz,zx] data tensor at elements
Beams	**Scalar**: A single value at beams
	Vector: A [x,y,z] data vector at beams

Each user-defined component must fall into one of the six classes in the second column above. It is not possible to mix classes in a single component.

Once a component has been defined it may be used exactly like "normal" data in any valid context within D3PLOT: data plotting, written value extraction, XY plotting, etc. Where vector and tensor components are defined standard operations such as extracting vector direction and magnitude, or deriving sub-components such as principal or von Mises sub-components, may be performed.

A user-defined component may have its name, content or calculation method changed at any time; however it may not be swapped between classes. (Although it may be deleted and re-created as a new class.)

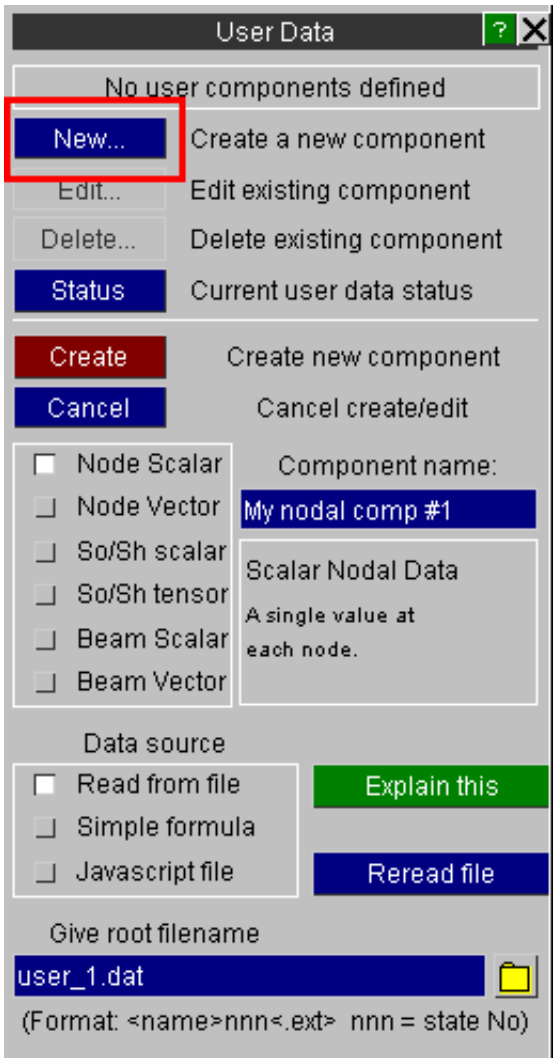
User-defined components are treated as "programme-wide" attributes, meaning that the same components apply to all models read into the database. If you need to apply separate components to different models it will be necessary to create new user-defined components with different attributes for the various models.

User-defined components must have names that are unique within D3PLOT. This means that not only must their names not overlap with each other, but they must also not be the same as "standard" components: for example you will not be permitted to create a user-defined component called "strain".

There are four types of user-defined component in D3PLOT. The first three are created interactively or externally, and the last is used only when using the Javascript interface. In summary they are:

(1) Read from file	Data is read from externally supplied ascii data file(s)	These three methods are easy to use, and in particular Simple Formulae may be created interactively. They are suitable for simple processing on a per node or element basis.
(2) Simple formula	Data is calculated internally from a "simple" formula using standard internal data and arithmetic.	
(3) Javascript file	Data is calculated using a small Javascript file.	
(4) User-defined Binary (UBIN)	Data is created via the Javascript interface (see section 11), not to be confused with method (3) above This is a much more powerful method than the three above, in that data can be extracted in an arbitrary fashion and written likewise, however it is not interactive. It is described briefly in section 6.4.18 below, and the API functions are described fully in the Javascript Interface Appendix.	

6.14.1 Creating a new user-defined component



The steps to be gone through are:

1. Use **New...** to start a new component definition

2. Choose one of the six possible data classes:

Node scalar	A single scalar value at nodes
Node vector	A [x,y,z] vector at nodes (3 values)
So/Sh scalar	A single scalar value at solids, shells, thick shells
So/Sh tensor	A [xx,yy,zz,xy,yz,zx] tensor at ditto (6 values)
Beam scalar	A single scalar value at beams
Beam vector	A [x,y,z] vector at beams (3 values)

1. Define a component name. A default name will be generated, but any name (up to 30 characters, an unique) may be used.

2. Define a data source from one of:

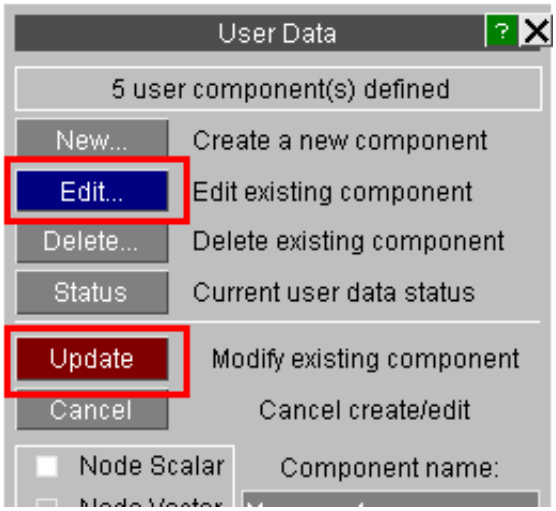
Read from file	Data is read from externally supplied ascii data file(s)
Simple formula	Data is calculated internally from a "simple" formula using standard internal data and arithmetic.
Javascript file	Data is calculated externally using a Javascript file.

1. Define a "root" data filename, simple formula or Javascript filename depending on the data source.

2. Finally **CREATE** the new component to save it.

There is also a category of "User Defined Binary Components", referred to as UBIN, that may only be generated by the separate Javascript interface to D3PLOT (not to be confused with the "Javascript file" data source referred to here). See the Javascript Interface documentation for more information about these.

6.14.2 Editing an existing user-defined component

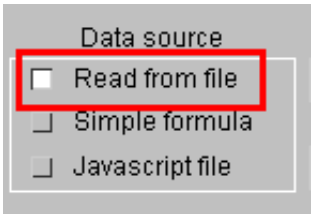


An existing component may be edited at any time.

- Click on **Edit...**
- Choose the component to be edited from the menu
- Change its attributes.
- Click on **Update** to save the changes

The component name and data source may be changed ad lib, but the basic class of the component (Node/SoSh/Beam and Scalar/Vector/Tensor) can not be altered. To change the component class it is necessary to delete it and then recreate it using the new class.

6.14.3 "Read from File"



This method assumes that data in the appropriate format has been generated externally and saved in ascii format data files.

Filename syntax

A separate file is required for each state, and its name is determined from the "root" filename supplied. This must be of the form

<*name*>nnn<*.ext*> Where:

<*name*>	Is an arbitrary prefix	For example a valid filename might be"nodal_1.dat" (decomposes to <**nodal_**>, <**1**>, <**.dat**>) The actual number used in the "root" filename is not material, it must just be a digit from 0 - 9.
nnn	Is one or more digits.	
<*.ext*>	Is an arbitrary extension	

For each state the digit <nnn> is replaced with the number of the state, so in the example above D3PLOT would look for the file sequence:

nodal_1.dat for state #1, **nodal_2.dat** for state#2, and so on.

The number **nnn** can have up to 10 leading zeros, so both **nodal_1.dat** and **nodal_0001.dat** would be acceptable filenames, and the number of leading zeros does not have to be the same for each state.

If a file for a given state is not found then values of 0.0 will be used for that state.

File content and structure.

In all files:
- Blank lines are ignored.
- Lines starting with #, % or $ are treated as comment lines and are ignored.
- Input is not case-sensitive: any mixture of upper and lower case can be used.
- Numeric input is "free": no particular field width or column number is implied.
- Field separators may be spaces, tabs or commas.
- Each row of data must be on a single line.
- Maximum line width is 256 characters.

The formats of the various data types are as follows:

Scalar data		
One of	**NODE SCALAR** **SOLID SCALAR** **BEAM SCALAR** **SHELL SCALAR** **TSHELL SCALAR**	A new header can appear at any time to start a new data category.
Optionally:	**DEFAULT** ***<value>***	A default of 0.0 is assumed if this line is not present
Optionally:	**SURFACE TOP** or **MIDDLE** or **BOTTOM** or **LAYER** ***<layer no>***	Ignored for Node, Solid or Beam data. For shells the current surface is assumed unless a SURFACE or LAYER line is defined.
Followed by any number of lines	***<label> <value>*** ***<label> <value>***	If <label> is not found in the model the line is ignored. Entities which don't have a value line are given the default value.

Example of a nodal scalar data file:
 	
`NODE SCALAR`
 	
`default 400.0`
 	
`12 1.382457E+02`
`13 -4.655358E+01`
`14 -2.706973E+02`
`15 -2.615501E+02`
`16 -1.364710E+02`
`17 8.553621E+00`
		Example of a shell scalar data file:
		
		`SHELL SCALAR`
		
		`default 10.0`
		
		`surface top`
		
		`1 1.23456e+02`
		`2 2.34578e-02`

Vector data		
One of	**NODE VECTOR** **BEAM VECTOR**	A new header can appear at any time to start a new data category. Vector data is not valid for solids, shells or thick shells
Optionally:	**DEFAULT *<3 values: x,y,z>***	A default of (0.0, 0.0, 0.0) is assumed if this line is not present
Followed by any number of lines	**<label> *<3 values: x,y,z>*** **<*label*> *<3 values: x,y,z>***	If <label> is not in the model found the line is ignored. Entities which don't have a value line are given the default vector value.

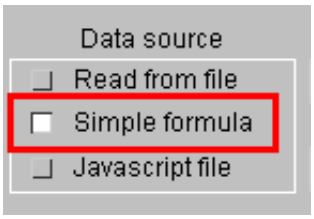
Example of a nodal vector data file:
 	
`NODE VECTOR`
 	
`default 1.0 2.0 3.0`
 	
`1 6.988275E+02 1.032284E+02 -1.468281E+02`
`2 7.038895E+02 7.875581E+01 -1.475109E+02`
`3 7.089514E+02 5.428316E+01 -1.481938E+02`
`4 7.140133E+02 2.981051E+01 -1.488766E+02`
`5 7.190753E+02 5.337874E+00 -1.495595E+02`
`6 7.241373E+02 -1.913478E+01 -1.502424E+02`
`7 7.291992E+02 -4.360743E+01 -1.509252E+02`
`8 7.342611E+02 -6.808007E+01 -1.516081E+02`
`9 7.393231E+02 -9.255272E+01 -1.522909E+02`
`10 7.375262E+02 -9.361953E+01 -1.273784E+02`		

Tensor data					
One of	**SOLID** **SHELL** **TSHELL**	Followed by one of	**TENSOR** **TENSOR_UPPER** **TENSOR_LOWER**	TENSOR implies linear data order TENSOR_UPPER implies upper triangular order TENSOR_LOWER implies lower triangular order (The tensor is implicitly symmetric)	[xx, yy, zz, xy, yz, zx] [xx, xy, xz, yx, yy, zz] [xx, yx, yy, zx, zy, zz]
Optionally:	**DEFAULT** ***<6 values>***			A default of [0.0, 0.0, 0.0, 0.0, 0.0, 0.0] is assumed if this line is not present If a default is supplied it must be in the order implied above.	
Optionally:	**SURFACE TOP** or **MIDDLE** or **BOTTOM** or **LAYER** ***<layer no>***			Ignored for Solid data. For shells the current surface is assumed unless a SURFACE or LAYER line is defined.	
Followed by any number of lines	***<label> <6 values>*** ***<label> <6 values>***			If <label> is not found in the model the line is ignored. Data must be in the tensor order implied by the header above Entities which don't have a value line are given the default values.	

Example of a nodal scalar data file:

```
SHELL TENSOR

Default 1.0 2.0 3.0 4.0 5.0 6.0
Layer 1

1 1.1567E+02 -3.9286E+01 -1.2514E-01 -1.2733E+01 3.8389E+00 -4.6773E+00
2 7.0594E+00 3.7640E+00 -1.1621E+00 5.0975E+01 9.0807E+00 -9.8825E-01
3 1.0105E+02 1.4159E+01 9.5627E-01 1.1344E+01 -3.2170E+00 -8.7537E+00
4 1.4473E+00 -5.2854E+01 1.7974E+00 2.5017E+01 -1.0200E+01 -5.3932E+00
5 8.9096E+01 -4.0171E+01 3.4451E-01 -4.2868E+00 3.0106E+00 -6.8030E+00
6 7.1164E+00 9.4949E+00 -1.1046E+00 4.2835E+01 4.2636E+00 1.6420E+00
7 2.4371E+02 2.0140E+01 -1.7336E+01 -7.3324E+01 -3.4727E+00 -1.1746E+01
8 2.1562E+02 3.8205E+01 2.8096E+00 -4.1887E+01 1.1632E+01 -1.0129E+01
9 2.2813E+02 2.3411E+01 1.8917E+01 -6.7636E+01 1.1483E+01 -1.0553E+01
10 1.9249E+02 -2.1300E+01 -1.3462E+00 -4.9531E+01 -5.8716E+00 -8.7624E+00
```

6.14.4 "Simple Formula"



Calculates results from a "simple" mathematical formula using built-in data components and standard maths operations.

All calculations are performed using floating point arithmetic and produce a single result, thus:
- Scalar components are evaluated from a single formula
- Vector components use three (independent) formulae: one for each of [x,y,z]
- Tensor components use six formulae, one for each of[xx,yy,zz,xy,yz,zx]

Syntax rules for Simple Formulae

Formulae follow typical "computer programming" syntax, using commonly understood operators.

Operators supported:		
*, /, +, -	Multiply, divide, plus and minus	* and / take precedence over + and -
** and ^	To the power of. Thus "x**2" is "x squared"	Note that "^" is Fortran syntax, ***not*** the C "exclusive OR" operator.
%	Modulus operator (x % y is remainder of x / y)	There is also a "mod" maths function.
(...)	Brackets may be nested to any (reasonable) level	(...) take precedence over all other operators
Note: all arithmetic is floating point, thus both "5.0 / 2.0" and "5 / 2" will both deliver the answer 2.5		

A formula must be a single line, up to 256 characters long.

Standard D3PLOT "built in" data components supported

Category	Acronym	Description	Acronym	Description
Nodal data	**bx** **by** **bz** **cx** **cy** **cz** **dx** **dy** **dz** **dm**	Basic X coordinate Basic Y coordinate Basic Z coordinate Current X coordinate Current Y coordinate Current Z coordinate X displacement Y displacement Z displacement Displacement magnitude	**vx** **vy** **vz** **vm** **ax** **ay** **az** **am** **temp**	X velocity Y velocity Z velocity Velocity magnitude X acceleration Y acceleration Z acceleration Acceleration magnitude Temperature
Solid and shell data	Stress and strain tensor-derived data is extracted in the global frame of reference unless the "**l**" (for "local") suffix is added to the component acronym, in which case it is extracted in the element local system. See below for how the Frame of Reference transformation applies to Simple formula components.			
Global tensor components	**sxx** **syy** **szz** **sxy** *or* **syx** **syz** *or* **szy** **szx** *or* **sxz**	XX stress YY stress ZZ stress XY stress (*symmetric*) YZ stress (*ditto*) ZX stress (*ditto*)	**exx** **eyy** **ezz** **exy** *or* **eyx** **eyz** *or* **ezy** **ezx** *or* **exz**	XX strain YY strain ZZ strain XY strain (*symmetric*) YZ strain (*ditto*) ZX strain (*ditto*)
Element Local tensor components	**sxxl** **syyl** **szzl** **sxyl** *or* **syxl** **syzl** *or* **szyl** **szxl** *or* **sxzl**	XX stress YY stress ZZ stress XY stress (*symmetric*) YZ stress (*ditto*) ZX stress (*ditto*)	**exxl** **eyyl** **ezzl** **exyl** *or* **eyxl** **eyzl** *or* **ezyl** **ezxl** *or* **exzl**	XX strain YY strain ZZ strain XY strain (*symmetric*) YZ strain (*ditto*) ZX strain (*ditto*)
Non-directional components derived from tensor data	**svon** **smax** **smid** **smin** **sav** **sms**	von Mises stress Max principal stress Mid principal stress Min principal stress Average stress (pressure) Max shear stress	**evon** **emax** **emid** **emin** **eav** **ems** **epl**	von Mises strain Max principal strain Mid principal strain Min principal strain Average strain (pressure) Max shear strain Effective plastic strain

Category	Acronym	Description	Acronym	Description
Shell only data	**rfx** **rfy** **rfxy** **rqx** **rqy** **rmx** **rmy** **rmxy**	Fx resultant force Fy resultant force Fxy resultant force Qx resultant force Qy resultant force Mx resultant moment My resultant moment Mxy resultant moment	**thk** **area** **eden** **hgen** **tstp** **mass** **madd**	Shell Thickness Shell Area Internal energy density Hourglass energy Timestep Mass Added mass
Solid only data	**erate**	Strain rate	**vol** **rvol**	Element Volume Element Relative volume
Beam only data	**bfx** **bfy** **bfz** **bfr** **bmxx** **bmyy** **bmzz** **brm** **bsxx** **bsyx** *or* **bsxy** **bszx** *or* **bszx** **bep** **beax**	Axial force YY shear force ZZ shear force Resultant force XX torsional moment YY bending moment ZZ bending moment Resultant moment XX axial stress YX shear stress (*symmetric*) ZX shear stress (*ditto*) Plastic strain Axial strain	**bsax** **bpe1** **bpe2** **bry1** **bry2** **brz1** **brz2** **bmy1** **bmy2** **bmz1** **bmz2** **baen** **bie** **brxx** **bbed** **baed** **bied**	Total axial strain Plastic energy at end 1 Plastic energy at end 2 Y rotation end 1 Y rotation end 2 Z rotation end 1 Z rotation end 2 Y moment end 1 Y moment end 2 Z moment end 1 Z moment end 2 Axial energy Internal energy Torsional rotation Bending energy density Axial energy density Internal energy density
"Extra" and ALE data	**sox***n* **shx***n*	Solid Extra component #n Shell Extra component #n	**ammg***n* **amss***n* **adens** **adomf**	Ale Multi-Matl group #n Mass of MM group #n Ale density Ale dominant fraction
Contact data	**csn** **cst** **csx** **csy**	Contact Normal Stress Contact Tangential stress Contact local X stress Contact local Y stress These are contact segment data averaged at nodes	**cfgx** **cfgy** **cfgz** **cflx** **cfly** **cflz** **cfm**	Contact global X force Contact global Y force Contact global Z force Contact local X force Contact local Y force Contact local Z force Contact force magnitude
User-defined components	**unos***n* **usss***n* **ubms***n*	Nodal scalar component #n So/Sh scalar component #n Beam scalar component #n	} Simple formulae may reference other user- } defined components as well as the standard } components above.	

Applicability of the "Standard" data components in the table above

The data components above are all implicitly available for nodes.

- Where the data is nodally-derived (eg current X coordinate **cx**) then the value is used directly.
- Where the data is element-derived (for example beam axial force **bfx**, or contact stress **csn**) then the nodal value will be the average of all elements of the relevant type meeting at that node.

Elements may only use components of their "native" type, or nodally-derived data.

- Where an element data component matches the element type (eg X stress **sxx** for shells, or YY bending moment **bmyy** for beams) then it is used directly.
- Where an element references a nodally-derived data component (eg X displacement **dx**) the value used will be the average of all nodes on that element.
- Where an element references an element-derived data component of a different type (eg a shell element refers to beam axial force **bfx**) then the result will be zero

Applicability of programme settings: Integration point, frame of reference, etc.

- Data is extracted "per integration point" for user-defined components, and it is not currently possible to specify data from some explicit integration point within a Simple Formula, meaning that you cannot use this method to assemble data from multiple integration points in an element.
- The current global / local / cylindrical "frame of reference" (see section 4.2.4.3) does **not apply** when a Simple Formula uses raw tensor components (eg Sxx) for solids and shells. The unqualified acronym (Sxx, Syz, ...) always extracts global data, and the acronym with an "l" suffix (eg Sxxl, Sxzl) always extracts element local data.

  However if you build a tensor Simple Formula (6 formulae) for solids or shells then the current "Frame of reference" **is** applied when it is used. In other words tensor Simple Formulae are treated as "raw data" once created, exactly like analysis data read from disk, and are subject to the same transformations. (Scalar and vector user-defined components are not transformed.)

Maths functions supported:

General functions	**sqrt(x)** **log(x)** (natural log) **log10(x)** (log base 10) **exp(x)** (e to the x)	**mod(x,y)** (modulo x / y) **max(x1,x2)** **min(x1,x2)**	**abs(x)** **int(x)** (truncate to integer) **nint(x)** (nearest integer) **ceil(x)** **floor(x)**
Trigonometric functions (use radians, not degrees)	**sin(x)** **cos(x)** **tan(x)**	**sec(x)** **csc(x)** **cot(x)**	**asin(x)** **acos(x)** **atan(x)** **atan2(x, y)**
Hyperbolic functions	**sinh(x)** **cosh(x)** **tanh(x)**		**asinh(x)** **acosh(x)** **atanh(x)**

.

Arithmetic types used.

ALL calculations in simple formulae are processed using double precision floating point arithmetic. This includes constants, and means that the "integer arithmetic" conventions of languages such as Fortran or C, and their associated truncation, do not apply.

For example the expression **`5 / 2`** is converted to **`5.0 / 2.0`**, and so returns the result **`2.5`**

Also the "integer" functions above (**`int, nint, ceil`**, **`floor`**) return integer values, but expressed as floating results.

For example **`nint(3.141592)`** evaluates to **`3.0`**

Examples of simple formulae

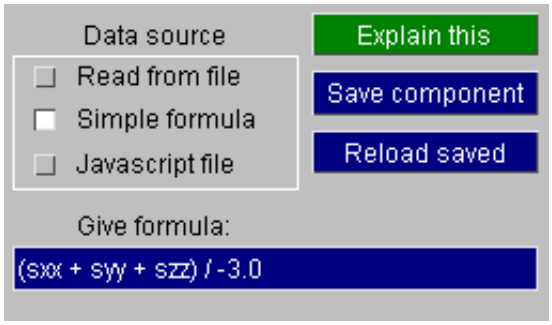
`(sxx + syy + szz) / -3.0`	Standard calculation for average stress
`1 / sqrt(2) * sqrt((sxx-syy)**2 + (syy-szz)**2 + (szz-sxx)**2 + 6*(sxy**2 + syz**2 + szx**2))`	Standard calculation for von Mises stress
`sxx*exx + syy*eyy + szz*szz + sxy*exy + syz*eyz + szx*ezx`	An estimate of strain energy density in a fully plastic section

Note that the "built in" data components described above can be referred to as simple acronyms, whereas maths functions require their arguments to be placed inside brackets.

Defining a scalar formula

In the case of a scalar component only a single formula is used.

In the example here the user is calculating pressure at nodes from stress components.

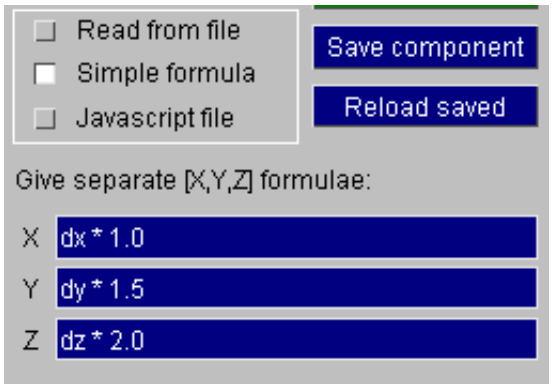


Defining a vector formula

In the case of a vector component three formulae are used, one for each of the [x,y,z] components. These are three wholly separate formulae which are evaluated independently.

In the example here the user is applying factors to displacements at nodes.

Note that each row must have a formula defined for the component to be valid.

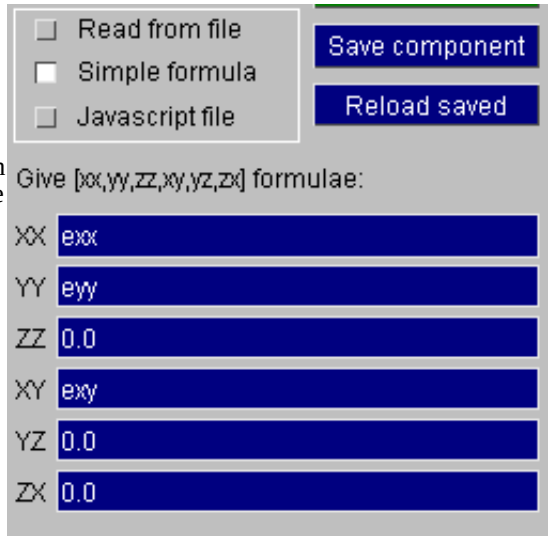


Defining a tensor formula

In the case of a tensor component six formulae are used, one for each of the [xx,yy,zz,xy,yz,zx] components. These are six wholly separate formulae which are evaluated independently.

In the example here the user is extracting only the in-plane (2D) components of strain for an element.

As with vectors each row must have a formula defined for the component to be valid even if, as here, some row values set explicitly to zero.



Further notes on Simple Formulae

Handling errors and missing data

If the expression being evaluated is grammatically legal, but not valid in a given context, then zero is returned. Typical examples are:
- If a formula references a built-in data component that is not present in the model database.
- If a mathematically illegal operation is attempted, such as divide by zero or the square root of a negative number.

Evaluation of element quantities at nodes

If an expression refers to an element data component, for example **sxx**, but is being evaluated for a node, then the relevant value is averaged from all legal elements at that node subject to the current averaging rules in force.

This can result in elements of more than one type being used: in this example component **sxx**, X stress, can be present in solids, shells and thick shells, so if multiple element types use the node then the value will contain contributions from all of them.

In general you should avoid contouring element quanties evaluated at nodes, since the effect is to smear out results and hence miss peak values at element centres. The result is identical to "low resolution" contouring of element data for scalar components.

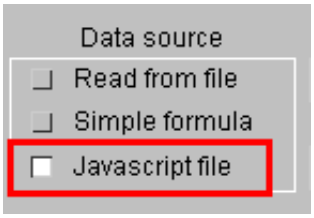
Evaluation of nodal quantities at elements.

If an expression refers to a nodal quantity, for example **cx** current X coordinate, but is being evaluated for an element, then the average value from all the nodes on that element is used.

Limitations of simple formulae

Simple formulae are powerful and very easy to use, but they do have limitations:
- They only operate on the current node or element, at the current surface or layer if relevant.
- They do not permit conditional expressions: you cannot use "if(...)" expression syntax

6.14.5 Javascript file



The third way of creating user-defined data components is via an externally defined Javascript file. Each node or element runs a short Javascript, the result of which becomes its value.

The Javascript capability in D3PLOT is still under development, but it currently provides the following:
- Access to the full range of "built in" data components, as supported by "simple formulae" above.
- Access to other element properties via further data functions.

Using these capabilities expressions can be evaluated in real-time outside D3PLOT, possibly with reference to external data, and then fed back in for plotting.

The Javascript file method is limited in that it can only work on the current node or element, however it does provide a bit more functionality than Simple Formulae in that conditional ("if") statements can be used making it possible, for example, to generate results from an internal lookup table.

For more complex processing it is recommended that you consider generating UBIN components using the main Javascript interface, see section 6.14.8 below for more information.

How the Javascript user components work

- You create a Javascript file (by convention ".js") outside D3PLOT using standard expressions, and also references to built-in data components.
- This is defined as the evaluation method for a user-defined component, currently limited to scalar components.
- It is test compiled to check for syntax errors, and if it is OK the compiled script is stored as the evaluation method for this component
- Thereafter nodes or elements run the compiled script on demand to obtain results.

The value used is the return value of the Javascript function. By convention this is the result of the final executable statement.

The (generally small) Javascript function is run for every element or node that requires data evaluation, so for a large model there may be some delay while all the values are computed.

Using built-in data components

The full range of data components used for simple formulae is available, with the following differences:

- Each component is a function in Javascript, not a variable. Therefore it uses the acronym in the simple formula table above followed by (..). For example:

  sxx, X stress, is referred to as **sxx()** inside Javascript. Note that the function has no arguments.

- Variable components above, eg "extra" values such as sox, pass the variable as the function argument. For example:

  sox3, Solid Extra component #3, is referred to as **sox(3)** inside Javascript

Using maths functions

Maths functions in Javascript are supplied by a "Math" class built into the language. For example:

Math.sin(x) evaluates sine (x)
Math.sqrt(y) evaluates the square root of (y)

Extra functions provided for Javascript

At present these are limited to:

label()	Returns the label of the item (eg node or element) currently being evaluated
pid()	Returns the part number of the element currently being evaluated. Zero is returned if called for an item that does not reference a part.
print(args)	Will write <args> to the controlling terminal.

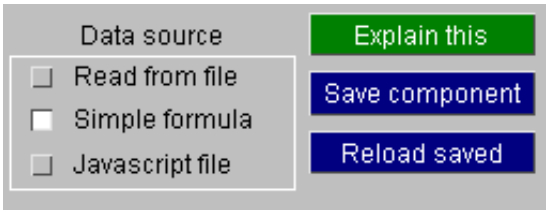
Examples of Javascript function

This function evaluates von Mises stress.

```
i = (sxx() - syy());
j = (syy() - szz());
k = (szz() - sxx());

l = Math.sqrt(i*i + j*j + k*k + 6 * (sxy()*sxy() + syz()*syz() +
szx()*szx()));

result = 1 / Math.sqrt(2) * l;
```

Those used to Fortran or C programming should note that Javascript is a very weakly typed language: numeric variables are "numbers", not integers, floats or other specific data types. Expressions are evaluated using double precision floating point arithmetic.

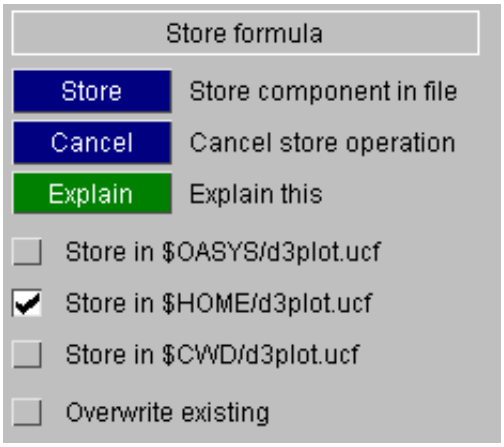
6.14.6 Saving and reloading user-defined components.



All types of user-defined component described above can be saved in a "User Component File" called "d3plot.ucf" using the **Save component** button, which will save the current component in the file for future reuse.

Likewise **Reload saved** will present a menu of components found in the "d3plot.ucf" file, and choosing one will import its settings for the component you are currently editing or creating.

Save component Saving the current component to file.



When you **Save** the current component you are asked to choose:

- Which file(s) to save it in.

  $OASYS is the system-wide file, usually where the software is installed.
  $HOME on Unix/Linux (**$USERPROFILE** on Windows) is your home directory
  $CWD is the current working directory

  The formula will be saved in all files selected for which you have write permission.

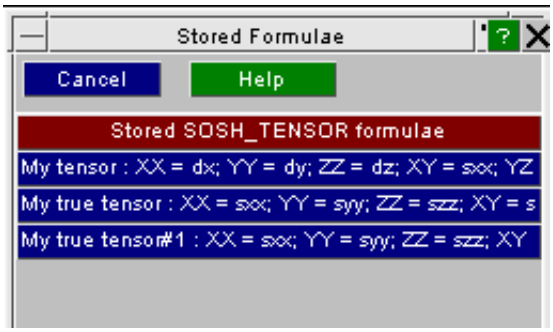
- **Overwrite existing** determines what happens if the name of the component to be stored matches one already in the file.

  If you choose to overwrite then the original component in the file will be overwritten with the new definition. Otherwise the current component's name will be added to the file, with its name modified by adding the suffix #1 (or #2, ...) to make it unique.

  Name comparison is not case-sensitive, but does consider embedded white space. For example:

  "**My beam component**" is treated as being identical to "**MY BEAM COMPONENT**"

Reload saved Reloading a component from disk to the current definition


Stored Formulae
Cancel
Help
Stored SOSH_TENSOR formulae
My tensor : XX = dx; YY = dy; ZZ = dz; XY = sxx; YZ
My true tensor : XX = sxx; YY = syy; ZZ = szz; XY = s
My true tensor#1 : XX = sxx; YY = syy; ZZ = szz; XY

When you choose to **Reload saved** the various "d3plot.ucf"files are scanned for components that match the current type, and these are listed in a menu for selection.

In this example the user is using a Simple Formula to define a solid/shell tensor, and there are three components to choose from.

Note that the component "My true tensor" has previously been stored twice, and because "overwrite" was not used the second definition has had "#1" added to it to make its name unique. (This is because D3PLOT requires components to have unique names, otherwise its internal parsing of components by name can get confused).

Automatic reloading of user-defined components.

When D3PLOT starts it automatically scans any "d3plot.ucf" files and loads any "unique" components, meaning that saved user-defined components automatically become available in a new session.

On start-up it scans the following directories:

$OASYS	Typically where the software is installed, and suitable for system-wide components. You may not have write permission to this directory
$HOME (Unix/Linux) **$USERPROFILE** (Windows)	Your home directory. This would be suitable for components you want to use for a range of projects.
$CWD (Current working directory)	The location of this varies: • On Unix/Linux it will normally be the directory from which you started D3PLOT • On Windows it will be the "start in" directory defined on an icon, unless over-ruled by a command-line "start-in" argument If you double-clicked on a file, or dragged a file onto the D3PLOT icon it will be the directory of that file.

In addition whenever a new model is opened any "d3plot.ucf" file in that model's directory is scanned, and any "unique" components are loaded.

Definition of a "unique" component

A component is deemed to be "unique" ***if***:

- Its name does not match any currently stored component.

  As explained above name matching is not case sensitive, but does consider embedded white space.

and

- Its data row(s) do not match stored data rows(s)

  FILE and **JAVA**script filename comparisons are made verbatim.

  FORMulae matching ignores both case and embedded white space. (ie the formulae are compressed to remove any white space before testing for matches)

If a component read from file is found not to be unique, ie it has already been read and stored, then it is ignored. In this way multiple reads of the same file, or the presence of a formula in more than one file, will not result in multiple definitions being read and stored.

If a component name matches, but its content (data rows) does not, then it is read in and its name has "#1, #2, etc" added to make it unique.

The Format of the "d3plot.ucf" file

Components are saved in the generic format:

<table>
<tr><td rowspan="3"><Type> <Source> <Name></td><td><Type> is one of</td><td>NODE_SCALAR
NODE_VECTOR</td><td>SOSH_SCALAR
SOSH_TENSOR</td><td>BEAM_SCALAR
BEAM_VECTOR</td></tr>
<tr><td><Source> is one of</td><td>FILE
FORM(ula)
JAVA(script)</td><td></td><td></td></tr>
<tr><td><Name></td><td colspan="3">Is a user-defined name up to 30 characters long</td></tr>
<tr><td rowspan="3"><Definition row #1>
: :
(Up to five further rows)</td><td>For FILE a single row</td><td colspan="3">A filename up to 256 characters long</td></tr>
<tr><td>For FORM 1, 3 or 6 formulae. Each on a new line, and each up to 256 characters long.</td><td>xxxx_SCALAR
xxxx_VECTOR
xxxx_TENSOR</td><td>one formula
three formulae
six formulae</td><td><scalar>
<x, y, z>
<xx,yy,zz,xy,yz,zx></td></tr>
<tr><td>For JAVA a single row</td><td colspan="3">A filename up to 256 characters long</td></tr>
</table>

The file is free-format ascii (text) and is not case sensitive.
Each row must be on a single line, up to 256 characters long.
Blank lines are ignored.
Lines starting with "%", "$" or "#" in column 1 are treated as comments, and are also ignored.

The only exception is that on Unix/Linux systems filenames ***are*** case-sensitive and they will be stored exactly as they have been defined.

The file may be hand-edited using a normal text editor. It will also be updated by the **Save** and **Reload** options described above.

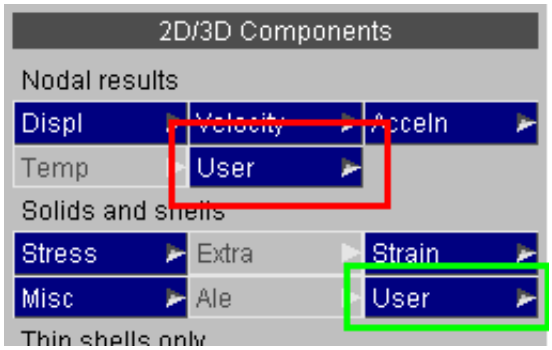
Here is an example of a typical file

```
$ Example file, created for user manual 1/11/2007

NODE_VECTOR FILE Example of nodal vector file
i:\demos\example1.dat

NODE_SCALAR FORM Example of scalar node form
sqrt(dx + dy + dz)

BEAM_VECTOR FORM Example of vector beam form
fx * eax
fy * eyz
fz * ezx

SOSH_TENSOR FORM Example of elem tensor form
sxx * exx
Syy * Eyy
sxx * ezz
sxy * exy * 0.5
syz * eyz * 0.5
szx * ezx * 0.5

BEAM_SCALAR JAVA Example of beam scalar Java
/local/demos/javascript/beam_scalar.js
```

6.14.7 Using user-defined data component

Once they have been created user-defined components become available for use in all contexts where "built-in" components may be used:

- 2D/3D plotting
- Beam plotting
- Vector plotting
- WRITE and XY_PLOT output

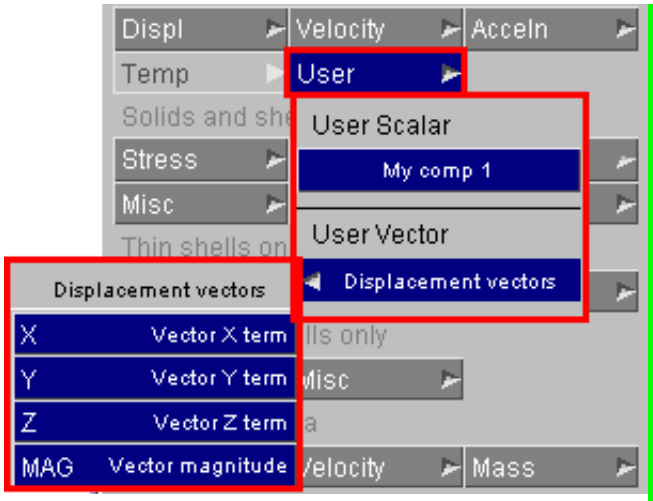


In 2D/3D plotting

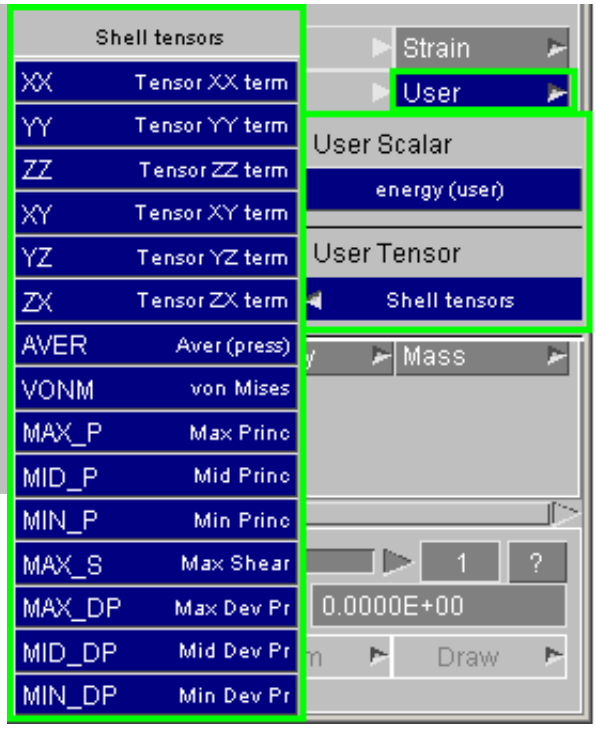
In this context the **User >** button(s) will become "live" in the **`Nodal results`** (outlined in red) and/or **`Solids and shells`** (outlined in green) contexts if components of the relevant types have been defined.

Under each category scalar and vector/tensor components are listed separately. Scalar components are selected directly, and the individual sub-components of the vector/tensor cases may be selected from popup sub-menus.

Selecting Nodal Vector sub-components



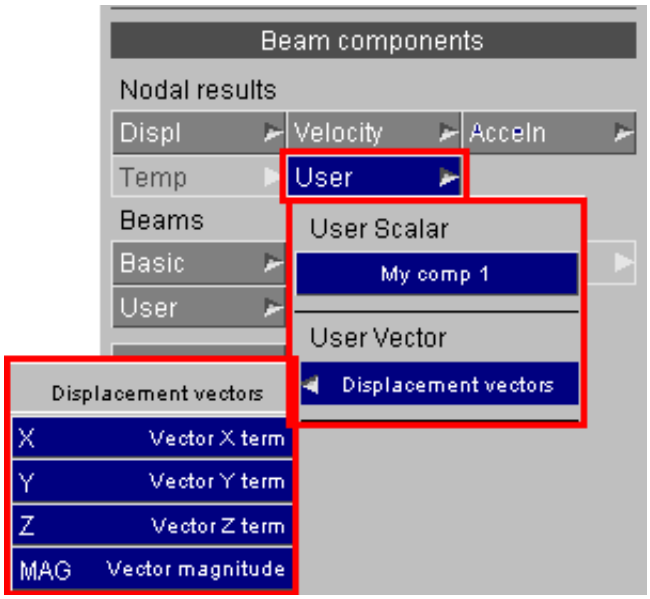
Selecting Element tensor sub-components



In Beam Plotting

As with 2D/3D plotting the User > button will become "live" if user-defined components for beams are created.

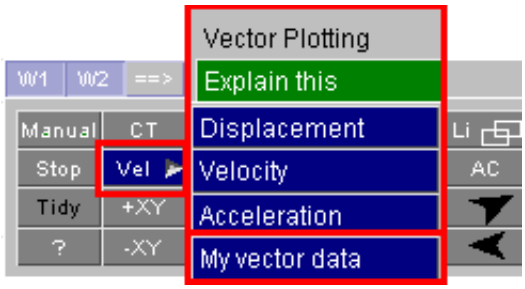
Sub-components for vector types are selectable in exactly the same way.



In Vector Plotting

In this context any Nodal Vector user-defined components are added to the standard list of components (Disp, Vel, Accel) eligible for vector mode (arrow) plotting.

Obviously in this context the vector component as whole is selected, as all three terms are used to generate the image.

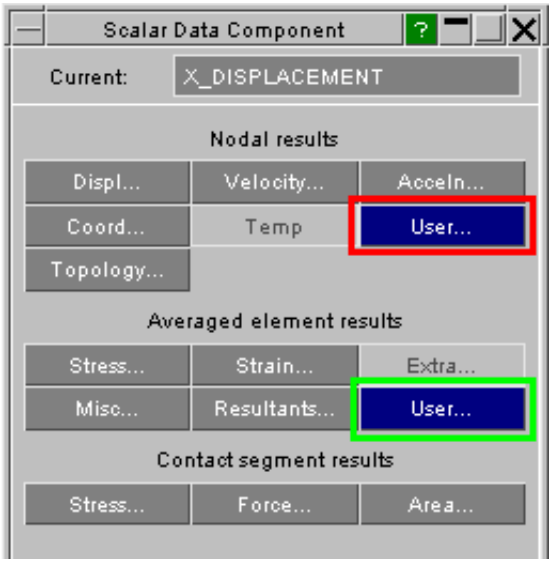


In **WRITE** and **XY_PLOT** panels

In these two contexts the **User...** button will become live in exactly the same way as described above if the relevant user-defined components have been defined.

The context shown here is for nodes, but the same applies to element-derived data.

The sub-menus give access to the top level user-defined components, and below that popup menus give access to the vector and tensor sub-components, again exactly as above.



6.14.8 "User Defined Binary" (UBIN) components generated from the Javascript interface

There is a fourth class of user-defined data component that shares all the attributes of those described above, which is used with the Javascript interface (described in section 11, and not to be confused with the "Javascript file" method above).

UBIN components are far more powerful than the methods above since data can be extracted for any node or element, from any state and at any integration point; and it can be written anywhere in the same way. In addition the Javscript API allows data to be read from and written to external disk files, so UBIN components can be created from any mix of internal and external data using algorithms of arbitrary complexity.

UBIN components are used in the same way as the three types above except that:

- UBIN components can only be created and manipulated via the Javascript interface.

- They are cached on disk in <jobname>.ubd files alonside the normal database files, meaning that they persist across D3PLOT sessions.
- Their data is stored in compact binary form, meaning that access is quick but <jobname>.ubd files are not editable.

Once a UBIN component has been created in the Javascript it will appear in the lists and menus of user-defined components, and can be used in exactly the same was as any of the other user-defined components described above.

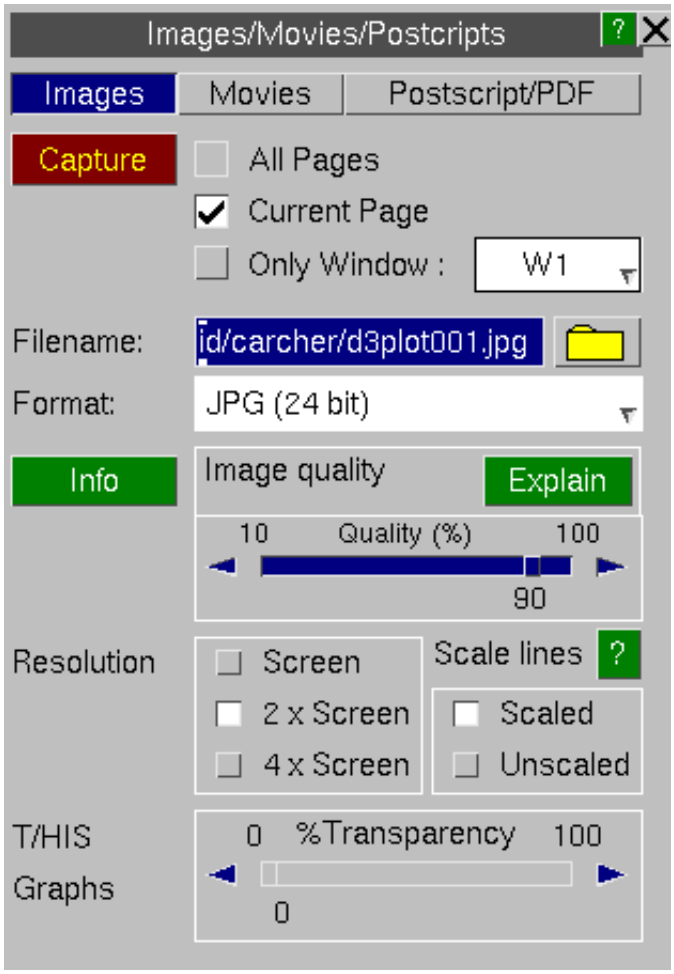
The creation and management of UBIN components is described in more detail in the Javascript Interface Appendix.

7 IMAGES

The **IMAGES** option allows static screen images and animations to be captured and also to be read in as background.

"Bitmap" static images and animations are handled using the menu below, whilst for Laser plotting see 7.3 LASER PLOTTING.

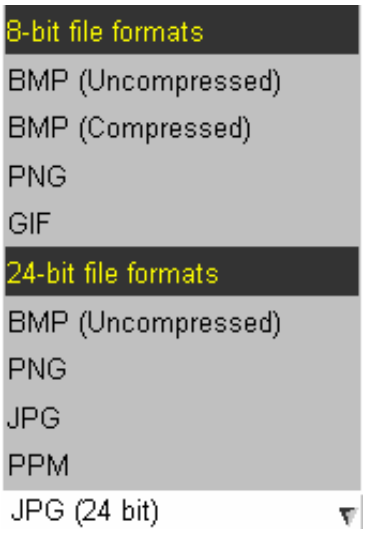
7.0 Creating static images and movies



CAPTURE

Captures a single static frame in one of the following formats:

8-bit file formats



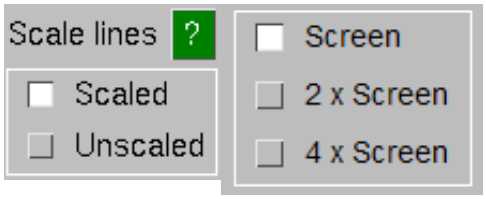
BMP Uncompressed	Uncompressed 8 bit Microsoft windows bitmap. The approximate size of the file (in bytes) is file size= image width * image height
BMP Compressed	8 bit RLE Microsoft windows bitmap.
PNG	8 bit Portable Network Graphics
GIF	Graphics Interchange Format

24-bit file formats

BMP	Uncompressed 24 bit Microsoft windows bitmap. The approximate size of the file (in bytes) is file size = 3 * image width *image height
PNG	24 bit Portable Network Graphics
JPG	JPEG (Joint Photographic Experts Group) file
PPM	Uncompressed **P**ortable **P**ix**M**ap. The approximate size of the file (in bytes) is file size = 3 * image width *image height

Various **.bmp** formats are available, and there are Controls for the dithering of the 8 bit-plane variants and palette optimisation .

RESOLUTION



All images can be output at either the screen resolution or at a resolution of either 2 or 4 times the screen resolution.

The widths of lines will appear to get narrower at 2 or 4 times the screen resolution. Scaling can be turned on to make the line widths match those on display.

CAPTURE

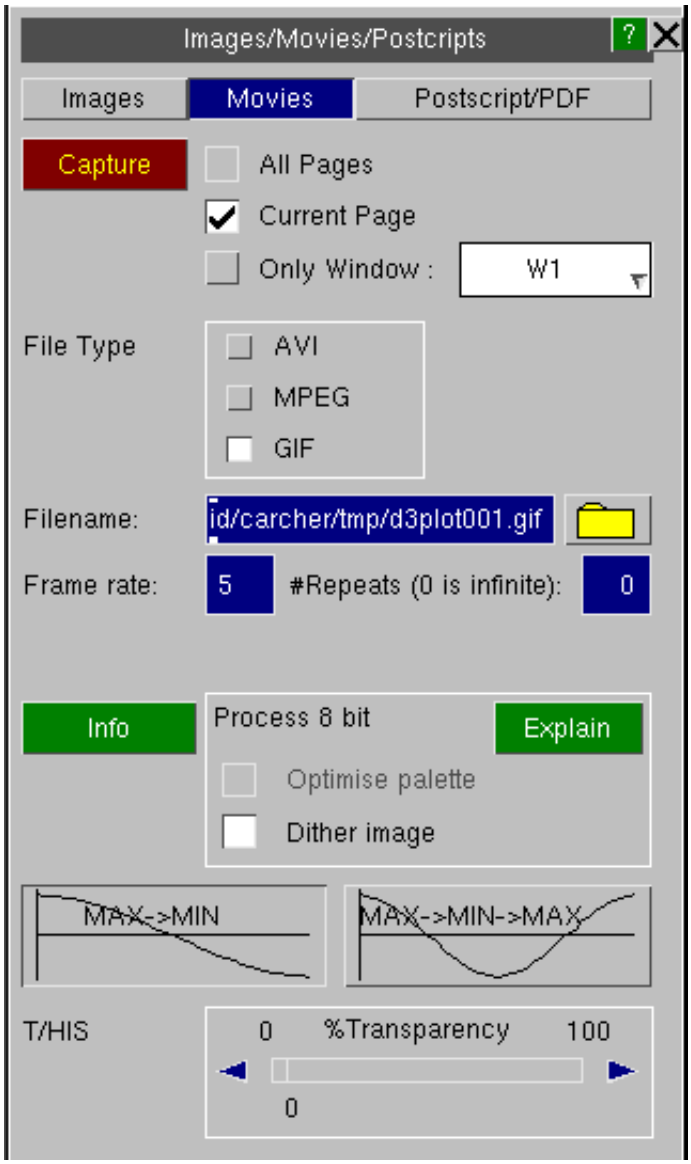
Captures the current animation in one of the following formats:

AVI **(.avi)**

MPEG **(.mpg)**

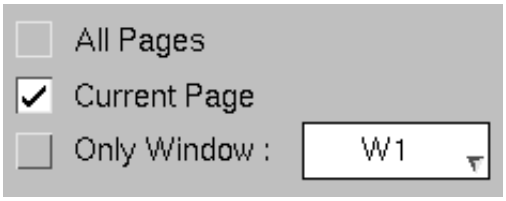
GIF **(.gif)**

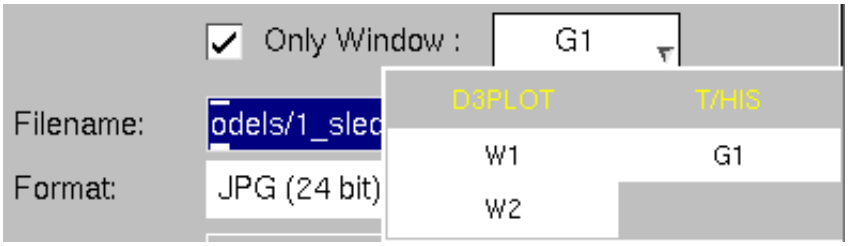
The file format, replay characteristics and frame repeat parameters can be defined.



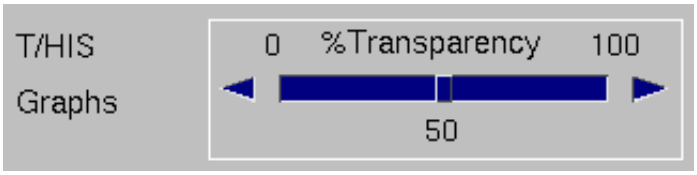
Multiple Windows

When multiple windows are present, any or all of these may be included in the images and animations by using the tabs and dropdown menu. If the T/HIS link is open the menu will have two columns; one for D3PLOT windows and one for T/HIS windows. The captured image will be the size of the window.





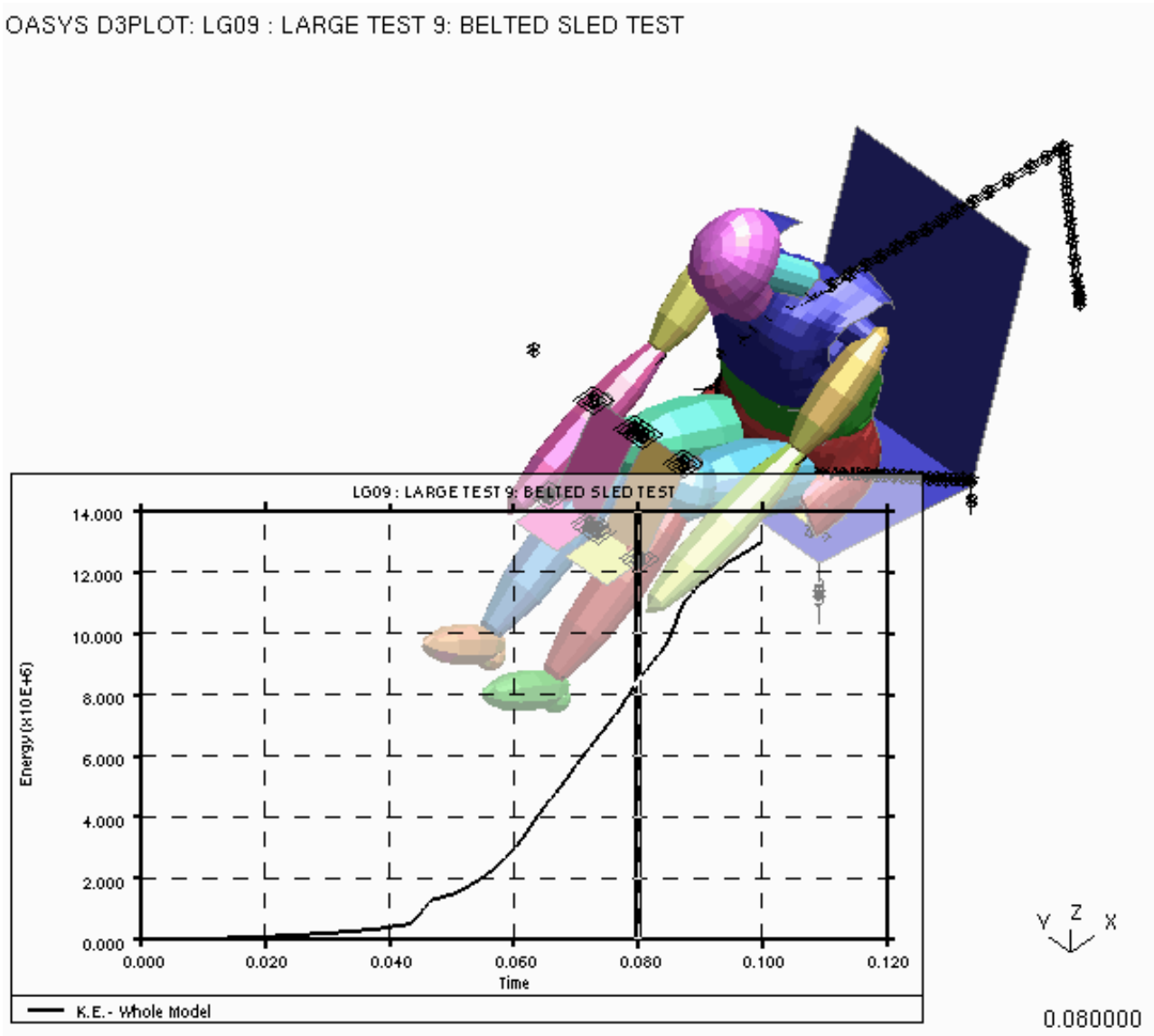
Capturing composite images of Linked T/HIS and D3PLOT Windows



When linked T/HIS is running it is possible to capture a composite image of both D3PLOT and T/HIS windows, and in addition windows may be made partially transparent. This is useful if the T/HIS window is a child of a D3PLOT window so that the underlying D3PLOT image can be seen.

The figure below shows an example of a 50% transparent "docked" T/HIS image overlying the D3PLOT one.

In the undocked or "sibling" cases a composite image will be the size of the rectangular bounding box required to enclose the selected windows.



7.1 Static file formats supported

JPEG	Joint Photographic Experts Group compressed format. This gives image quality nearly comparable to 24 bit-plane bitmaps, but with a file of < 5% the equivalent size. JPEG format is supported by all common visualisation packages and is recommended for all applications unless image quality is of paramount importance.
8 Bit Compressed BMP :	8 bit runlength encoded (RLE) Microsoft windows bitmap.
8 Bit Uncompressed BMP :	8 bit uncompressed Microsoft windows bitmap. The approximate size of the file is [image width * image height] bytes.
24 Bit Uncompressed BMP:	24 bit uncompressed Microsoft windows bitmap. The approximate filesize is [3 * image width * image height] bytes.
PNG:	24 bit lossless compressed **P**ortable **N**etwork **G**raphics image. PNG offers the similar degree of compression as GIF but has better colour quality.
GIF:	8 bit lossless compressed **G**raphics **I**nterchange **F**ormat.
PPM:	24 bit uncompressed **P**ortable **P**ix**M**ap. The approximate size of the file is [3 * image width *image height] bytes.

7.1.1 Controls on the quality of 8 bit-plane bitmap files.

24 bit BMP files tend to be huge, and the space saved by using the compressed 8 bit format is attractive. The trouble is that without further processing the image quality obtained when 24 bit images are truncated to 8 bits is definitely not!

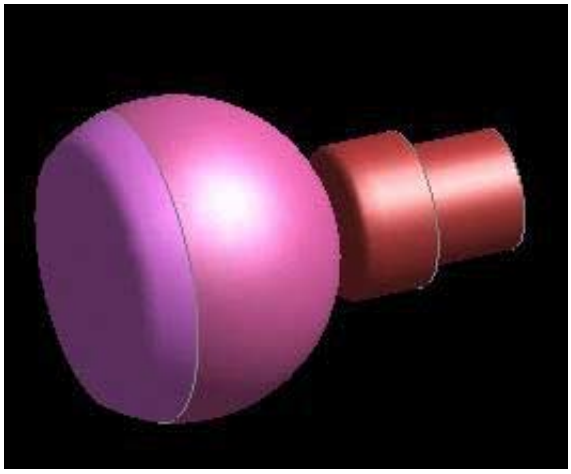
The problem is the number of colours available in the 8 bit format is 2**8 = 256, and this gives rise to "banding" when the least significant bits of the original colour definitions are lost as the original 16 million colours are truncated.

Dither image

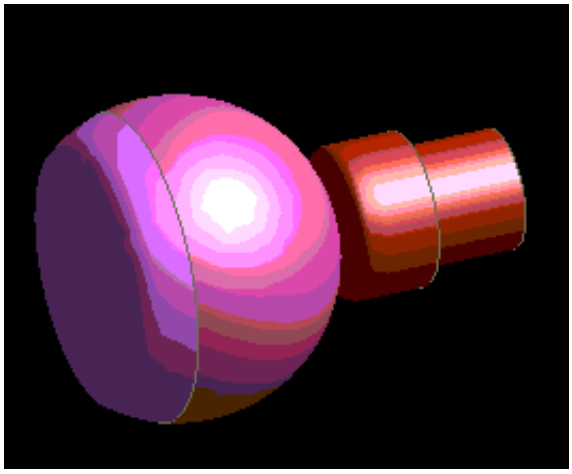
The following sequence of images show how the different levels of dithering affect the quality and file size of a compressed 8 bit image, comparing it with the JPEG equivalent.

For static images there is no advantage in using BMP files over JPEGs: they are larger and of inferior quality. However if you are unable to use MPEG animation, and have to revert to AVI format, the various permutations of image quality and filesize below will be of interest when trying to obtain the best compromise between image quality and overall file size.

The ideal would be an AVI file composed of JPEGs, or MJPEG format. Sadly all such formats are proprietary and, perhaps as a consequence, are not supported by the typical players currently available. Hopefully this will change in the future.



Here is the original 24 bit-plane image, saved as a JPEG file. **Size 5.1kB**



This is the undithered equivalent bitmap image. **Size 7.3kB**.

Note how the discretising affect of mapping onto a limited colour palette has caused "banding" which makes the image almost unusable. However at least the files are small!

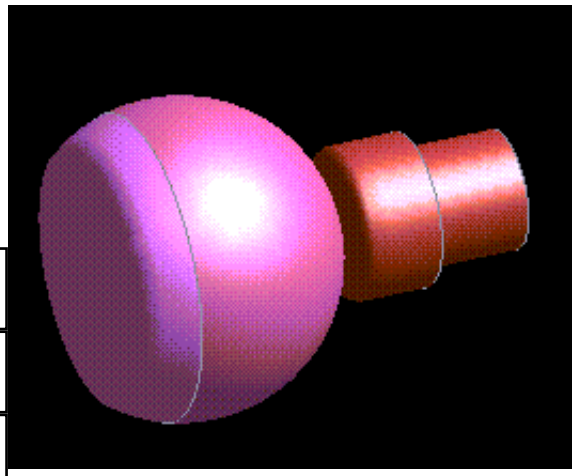
"Dithering" is a technique in which an ordered pattern of noise, **xxxxx** in the table below, is added to the least significant bits of a colour value to make it alternate between two adjacent shades.

Consider the bits for a single colour in this image that have been truncated from 24 bits (8 bits each of Red, Green and Blue). Truncated bits are shown in lower case.

Original Red byte **001?????**	truncates to	**001ooooo**
Adding the dither pattern **oooxxxxx** to the bottom 5 bits of the original byte gives		
001????? + **oooxxxxx**	giving either or	001ooooo 010ooooo

The result may be truncated to **001ooooo**, or the increased to the next shade up **010ooooo**, depending on the trailing bits **?????** and the noise value **xxxxx** at that pixel.

The effect is to produce a composite shade that is somewhere between the two originals.



Here is a dithered version of the image above, **size 8kB**.

Some banding is still visible, but it has been reduced to an acceptable level, and the coarsening of spatial resolution is also evident.

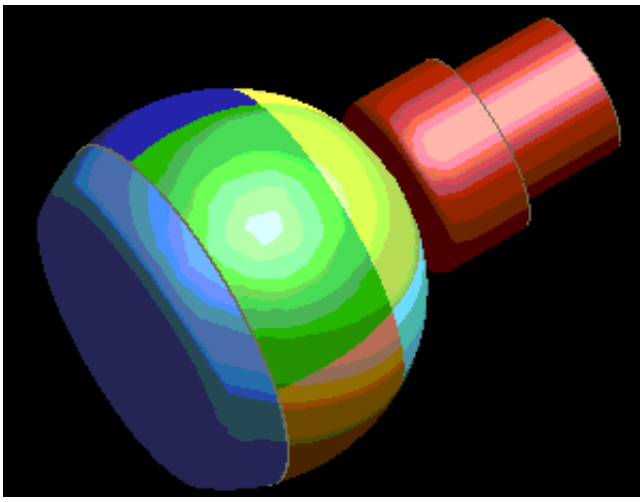
Generally dithering gives the best results for animated 8 bit images

Palette Optimization

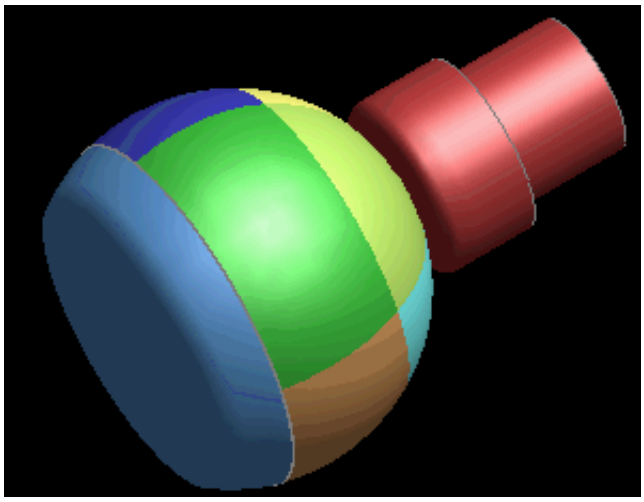
When 8 bit images are produced the 24 bit palette has to be reduced to only 256 colours. To do this the best way is to use Palette Optimization to choose the most representative colours used in the image.

Without Palette Optimization 256 colours can be chosen uniformly along the original 24 bit palette, missing out important colours.

The following figures show the differences in images with Palette Optimization.



This is the original image, saved as a GIF, with no dithering or palette optimization. **Size 6.1kB**

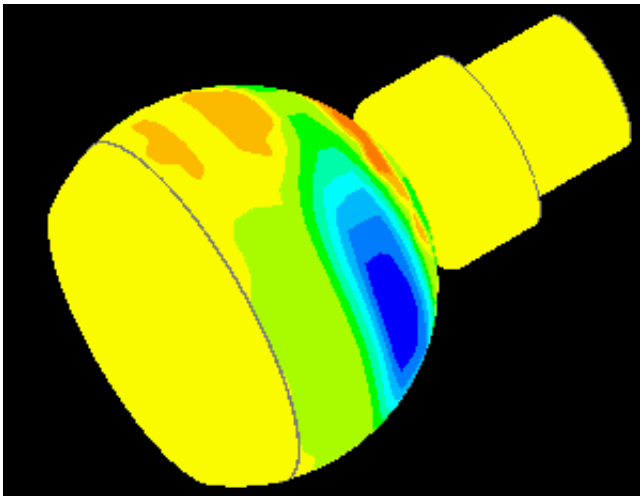


This is the image, saved as a GIF, with palette optimization. **Size 12.6kB**

Note that whilst there are still bands, the coarseness of them has diminished.

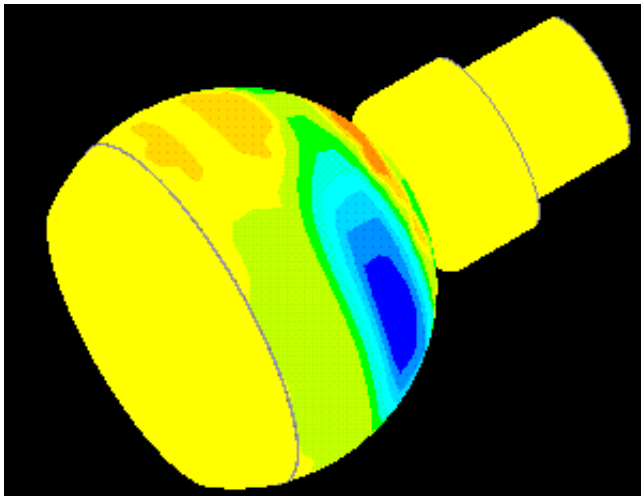
The two images above were created from a shaded image plot in D3PLOT and therefore contained a lot of different colours. The banding is present because the palette has been reduced to the 256 most representative colours in the image.

The following figures are images of a contour plot in D3PLOT.



This is the image, saved as a GIF, with palette optimization. **Size 3.1kB**

Note that due to the smaller number of colours in the image, there is no banding.



This is the image, saved as a GIF, with dithering. **Size 3.7kB**

Dithering is not well suited to images with distinct colours since by its nature it produces colours that are somewhere in between neighbouring colours. This is effective with shaded images, but not with images where there are sharp changes of colour.

7.2 Animation file formats supported and their attributes

D3PLOT supports three animation formats:

AVI (**.avi**)	**Adapted Video for Internet.** This format acts as a "wrapper" around a sequence of static images, adding information about their content and the replay rate required. In principal any still image format can be wrapped in this way, but in practice commonly available players only support a limited range of formats; and the coder/encoder ("codec") software for higher performance formats tends to proprietary. For example the Indeo, Cinepak and Sorenson encodings are all copyrighted and expensive to obtain. D3plot supports MJPG(Motion JPEG) encoding from version 9.3 which gives great compression, and each frame gets good image quality which is the same as a JPEG. D3PLOT supports also bitmap encoding which, while it does not give good compression or image quality (unless the prohibitively voluminous 24 bit option is used), is at least in the public domain and will guarantee to play back in any package. The dithering options (see **Controls**) that have been added in version 8.2 give a reasonable compromise between image quality and file size when 8 bit compressed bitmaps are used.
MPEG (**.mpg**)	**Motion Picture Experts Group level 1 encoding.** This is a format with variable compression in which quality (in %) can be traded off against file size. In D3PLOT the default quality is set to 90%, which gives results slightly inferior to the JPEG files. In normal video streaming an MPEG video is designed to mix "I" frames, containing full information, with several "P" frames, containing only the difference between this and the preceding "I" frame. This is based on the assumption that "real life" scenes don't actually change much between successive frames, and sending only "difference" information in the "P" frames saves a lot of file space. D3PLOT generates only "I" frames since the differences between successive frames are large. The format is designed to play back at video speed of 25 frames/second, and hence it places quite severe demands on the machine doing the playback. It is also limited to a window size of 768 x 576 pixels according to the MPEG-1 standard. If either of these limitations pose a problem it may be necessary to revert to AVI files. Our experience has shown that a Pentium III processor of at least 500MHz is necessary to obtain satisfactory replay under Windows. Moreover playback is better under the more recent versions of software released by Microsoft, running under Windows 2000 or equivalent.
GIF (**.gif**)	**Graphics Interchange Format.** This format acts as a "wrapper" around a sequence of static GIF images, adding information about their content and the replay rate required. GIF animations will tend to be smaller than AVIs and MPEGs and have the advantage that they can be inserted into Powerpoint as images. This means that unlike AVIs and MPEGs the files do not need to be carried separately with the presentation.

7.2.1 FRAME_RATE Frame rate when played back

This option can be used to specify the desired playback speed in frames/second.

The playback speed of an AVI movie is encoded into the AVI file along with the length of the movie (seconds). If too high a playback speed is requested then most AVI movie players will skip some frames to ensure that the movie plays for the correct length of time.

The playback speed of a GIF animation is also encoded into the GIF file.

At present the frame rate of an MPEG movie cannot be controlled: the MPEG standard fixes this at 24, 25, 30 or 60 frames/second - we use 25 fps.

7.2.2 #REPEAT Number of repeats when played back

This option can be used to make some movie formats repeat themselves. The details vary by format as follows:

Repeating AVI files

Some AVI movie players contain an option to play a movie more then once, and if your player supports this option you should use it for repeats since the AVI format is extremely inefficient in this respect: it requires <#frames x #repeats> to be added to the file, which can make it very large.

However if your AVI movie player does not provide such an option then use **#REPEAT** to make the movie play more than once. If 0 repeats are entered for "infinite" replay 32 repeats only will be encoded because of the file size issue described above. If you ***must*** have more repeats then enter an explicit number in the range 1 to 65536, although you should consider using animated GIFs instead, as these will loop any number of times with no file size penalty.

Repeating animated GIF files

GIF movies can also be made to repeat from 1 to 65535 of times, or if 0 repeats are entered the animation will play indefinitely. There is no file size penalty for specifying repeats.

Repeating MPEG files

MPEG, being designed for video streams, does not include the concept of repeated looping back from the start so the "**repeat**" option is not available. However most players will allow you to cycle repeatedly through MPEG files.

7.2.3 **FORMAT** AVI file formats supported

AVI movies can be written using a wide range of file formats. D3PLOT supports the following four, since they have been found to play successfully using 'xanim' on a range of UNIX machines and Microsoft's ActiveMovie on PC's.

MJPG:	Each frame within the movie is stored as an 24 bit JPEG image. This format offers the best combination of quality and size and is recommended unless your player will not support it.
8 Bit Compressed :	Each frame within the movie is stored as an 8 bit RLE Microsoft windows bitmap.
8 Bit Uncompressed :	Each frame within the movie is stored as an 8 bit uncompressed Microsoft windows bitmap. The approximate size of the movie is: [#frames * image width * image height] bytes
24 Bit Uncompressed :	Each frame within the movie is stored as an 24 bit uncompressed Microsoft windows bitmap. The approximate size of the movie is: [3 * #frames * image width * image height] bytes

The **Controls** apply equally to the 8 bit formats here in exactly the same way as they do to static images.

7.2.4 **QUALITY** The playback quality of MPEG files.

MPEG compression is "lossy", and the degree of loss is defined via the quality setting in %: lower quality gives smaller files, but the image quality degrades noticeably.

In D3PLOT the default quality is set to 90%, which gives an image quality marginally inferior to static JPEG images, but definitely superior to the best 8 bit AVI formats. File size will be broadly equivalent to the size of the JPEG x the number of frames in the animation.

(Quality does not apply to AVI or GIF files.)

7.2.5 **MAX->MIN** or **MAX->MIN->MAX**

Most AVI players are able to generate a 360^0 modeshape animation from 180^0 worth of frames, (the **MAX->MIN** case). For those which are not use **MAX->MIN->MAX** which duplicates the frames to produce a full 360^0 worth.

Next section

7.3 LASER PLOTTING

7.3.0 Introduction to Laser Plotting

By default all graphics images generated by D3PLOT are sent only to the screen, but you can choose to copy them to laser files (postscript and pdf files for a laser printer).

This is done by pressing the "Plot" button when you are in Postscripts/PDF.

7.3.0.1 Laser language and file format used.

At present D3PLOT writes Postscript laser files, using PS ADOBE level 2.0 commands, and PDF files. These are ASCII files that can be viewed and edited using any common editor.

"Encapsulated" Postscript files are not written, but later in Section 7.3.4 the very simple edits required to convert a file to encapsulated form are given.

Laser output is switchable between A4 (297 x 210mm), A3(420 x 297mm) and US "letter" (11" x 8.5") paper sizes. However the Postscript language makes it easy to edit files to fit other sizes.

The laser driver defaults to "PDF" file format, you can opt for "Postscript" laser file.

7.3.0.2 Number and orientation of plots on a page.

The laser driver defaults to "landscape" orientation, with one plot per page. You can opt for "portrait" orientation and, in both cases, put multiple plots on a page in a variety of layouts.

7.3.0.3 Resolution setting.

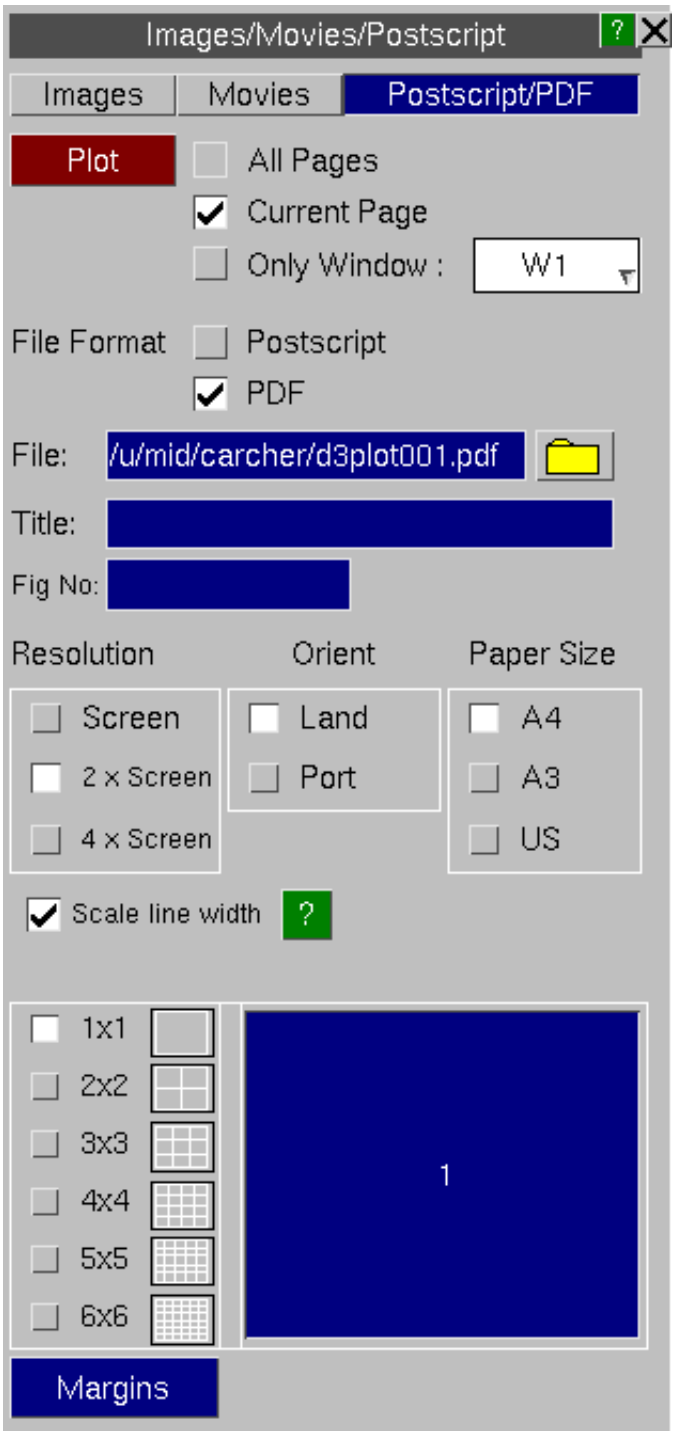
Postscript and pdf files generated can have higher than current screen resolution. Screen resolution, twice and four times screen resolutions are all available.

7.3.1 **LASER** Controlling laser plotting using the Laser Plotting panel.

This figure shows the basic laser plotting panel.

This is invoked by the Postscript/pdf command under Images->Write in the top menu box.

It both controls and shows the status of the current laser file (if any).



7.3.1.1 Plot button

Press "Plot" button when you want to plot the current view on the screen into a postscript or pdf file.

Any plot directed to laser file is sent by default to the next free sub-image (if the file has multiple plots per page), or file (if only a single image per file, or the multiple page is full).

When multiple sub-images in a file are in use the next image to be written is shown by depressing the appropriate icon in the file layout panel. You can override this and choose a different sub-image: see Section 7.3.1.5 below.

7.3.1.2 Choosing the laser filename

When no file is currently in use the **File:** entry box will be available. You can give any valid filename for the next laser file to be written, or let D3PLOT choose one for you. You can also use the button to select a file via the standard file filter box.

If the file already exists you will be queried to check that you genuinely want to overwrite it: you cannot append to existing laser files.

The default naming convention used by D3PLOT for laser files is **`postNNN.pdf`**, where:

`NNN` is a 3 digit number (with leading zeros if required) in the range **`001 - 999`**.

Any existing files are skipped when the next file in the sequence is computed, so in the example above file **`post001.ps`** already exists.

7.3.1.3 Defining a label and figure number for laser plots.

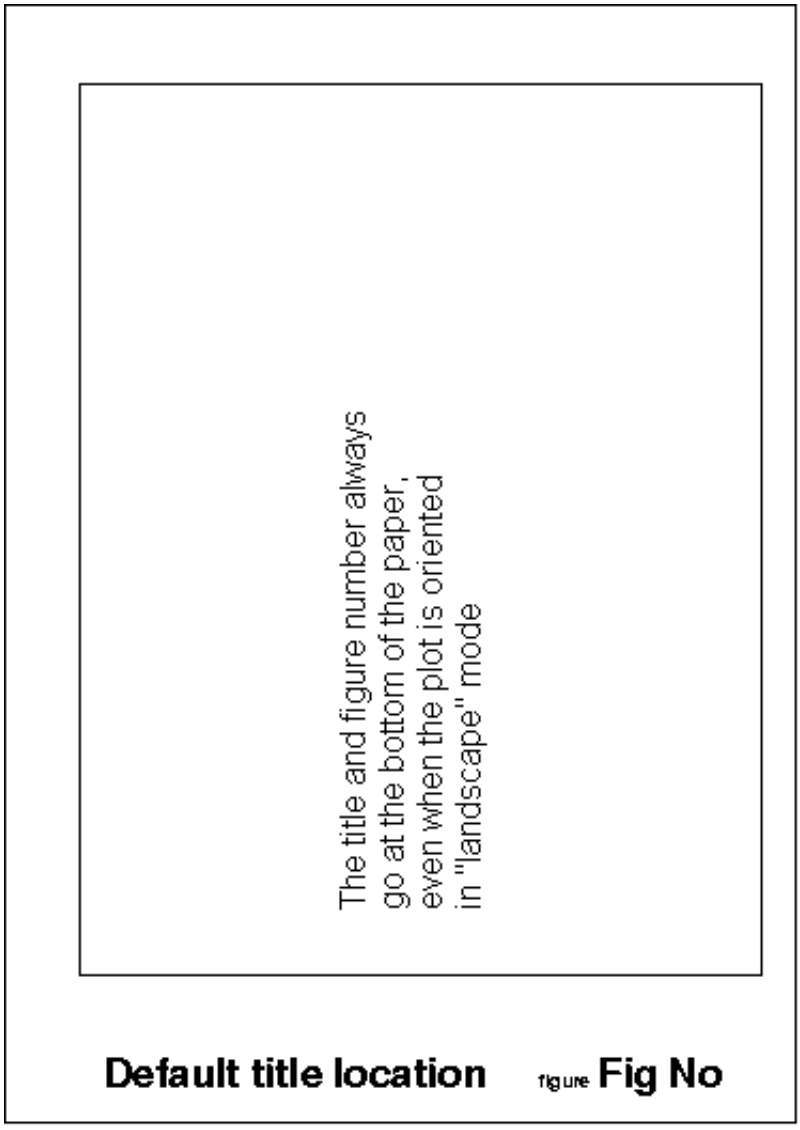
By default laser files are not labelled and have no figure number, but you may add either or both of these. They are always put at the bottom of the page, along the short edge, regardless of the orientation used for plots.

This figure shows the standard locations for title and figure number on laser plots.

The title may be up to 80 characters long, and is split over two lines if necessary by D3PLOT.

The figure number may be any string (not just a number), and is preceded by the word "figure". It is suggested that it is 6 characters or less long: here "12a" was used.

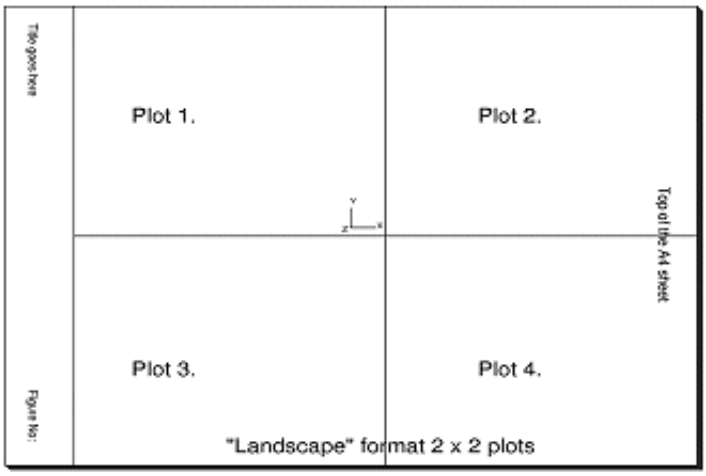
This plot is written in "landscape" format, and reinforces the point that the title and figure number always go at the bottom of the paper, regardless of the orientation of the plot contents.

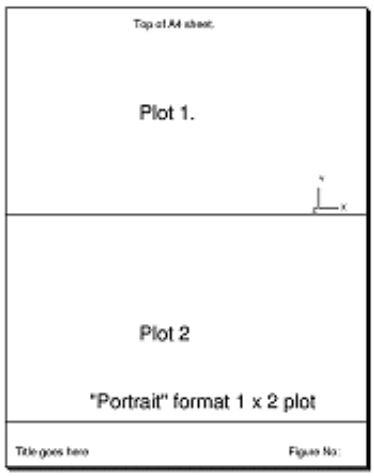

The title and figure number always
go at the bottom of the paper,
even when the plot is oriented
in "landscape" mode
Default title location
figure Fig No

7.3.1.4 **Orientation** Setting Landscape or Portrait plot orientation.

By default plots are in "Landscape" orientation, with the long side of the plot aligned with the long side of the paper, but you can choose "Portrait" format instead.

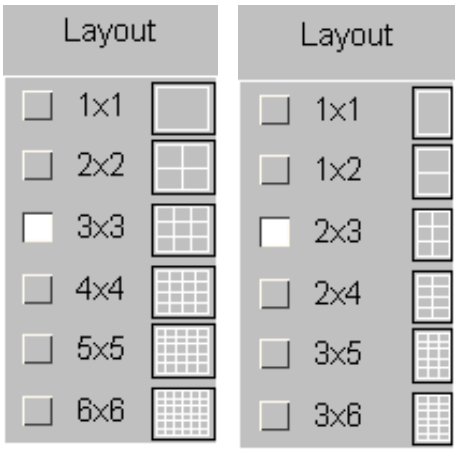
The figure below shows examples of both landscape and portrait format plots, showing how they are aligned on the paper.





This example shows examples of Landscape and Portrait plots, showing how they are oriented on the paper.

7.3.1.5 Layout Controlling the number and layout of sub-images.



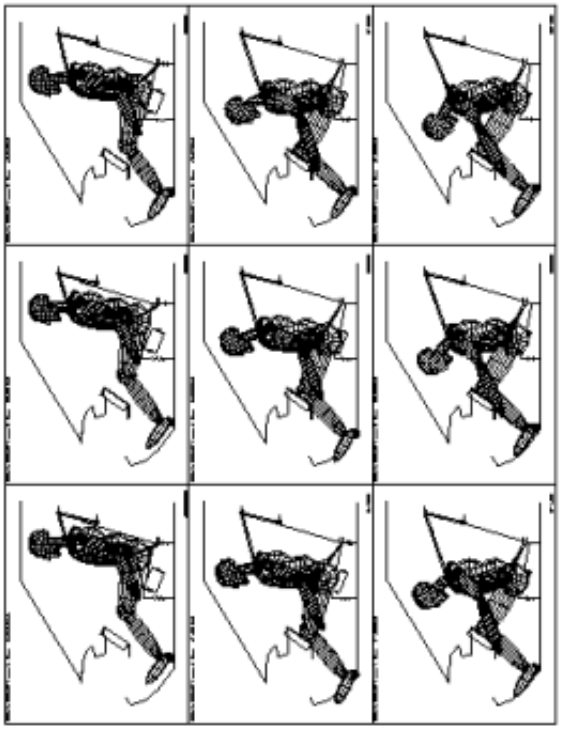
LANDSCAPE PORTRAIT

In both landscape and portrait formats it is possible to have more than one plot on a page.

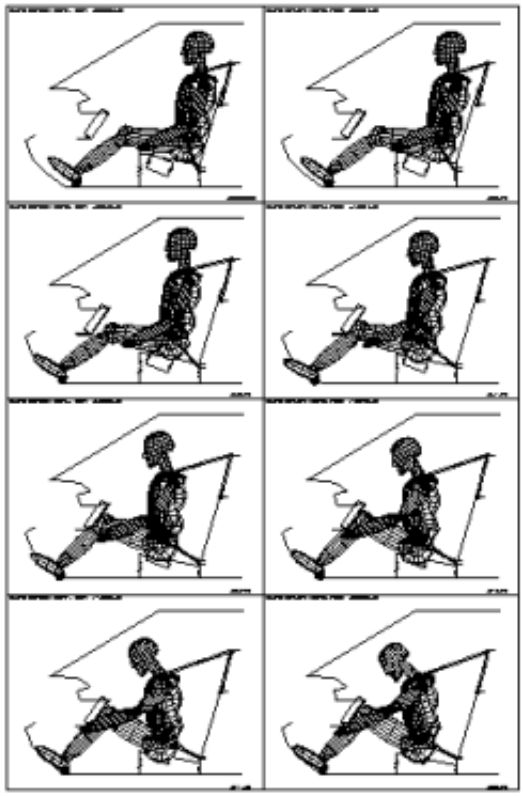
Various pre-programmed permutations of <#x> x <#y> plots are available as shown here.

Each individual plot on a page will be referred to from now as a "sub-image".

The figures below show examples of 3x3 Landscape and 2x4 Portrait multiple plots.



EXAMPLE OF 3 x 3 LANDSCAPE OUTPUT figure **7.1.5a**



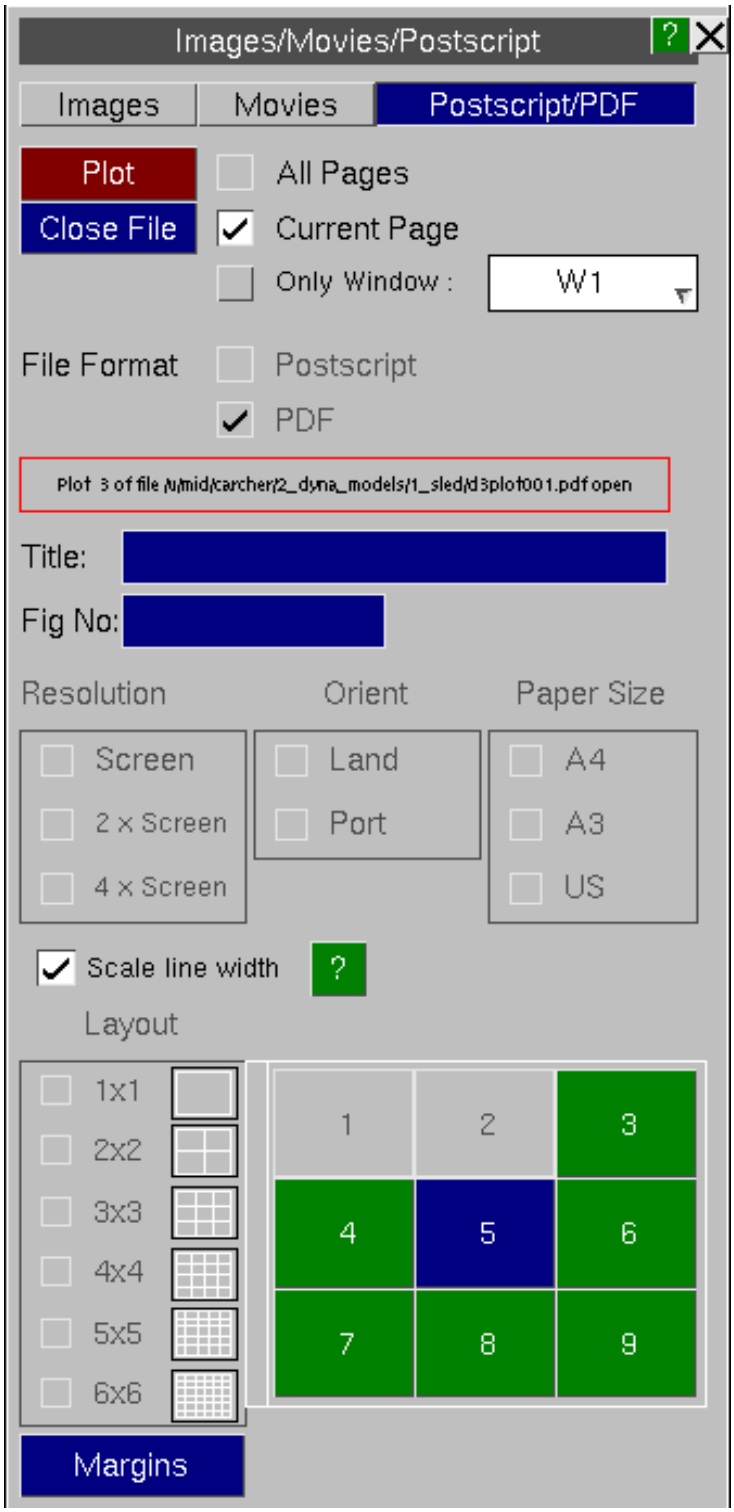
EXAMPLE OF 2 x 4 PORTRAIT OUTPUT figure **7.1.5b**

Controlling the order in which multiple plots are drawn.

The right hand menu shows a typical laser panel for a 3x3 portrait plot in which sub-images 1 and 2 are complete.

Normally sub-images are written in the order #1 to #n, but if the user wanted the next plot to be drawn to sub-image #5 instead of #3, he would click on the [5] icon where the button gets coloured in blue instead of the [3] icon as it normally would.

Next sub-image would be the next free one, i.e. #3 to receive the next plot. The [3] icon will be coloured in blue.



The status of files, and sub-images within files.

D3PLOT laser files, and sub-images within files, have one of three possible states.

Inactive	**Green**	No graphics written yet, and not selected for the next plot.
Selected	**Blue**	No graphics written yet, but selected to receive the next plot.
Closed	Greyed out	File/sub-image complete, and cannot receive any more information.

The colours referred to above are used for the button icons on multiple sub-image panels, as shown in the figure above. Only green icons (ie those which are currently inactive) may be selected to receive the next image.

How sub-image status affects the destination of graphics.

(1) If no graphics have been written to a sub-image then the next plotting command will send laser output to the the sub-image currently "selected".

By default this will be the lowest numbered sub-image that has not yet been written to, but you can choose another as described above.

(2) Once graphics have been sent to the sub-image its status changes to "closed" This means that it cannot receive further graphics.

Interaction between sub-images and files

A file with only a single image in it is treated in exactly the same way as an individual sub-image above, except that it is (implicitly) always "selected" for plotting until something is drawn in it.

A file with sub-images remains current (ie open) until all of the sub-images in it have been "closed", or the user closes it prematurely with a **CLOSE FILE** command. Then D3PLOT defaults to the next default filename as defined in Section 7.3.1.2 above.

The importance of closing files.

While a file is still current it is still connected to the programme, and at least some of its contents will still be held in system buffers. If you want to send it to a printer you must close it first using a **CLOSE FILE** command.

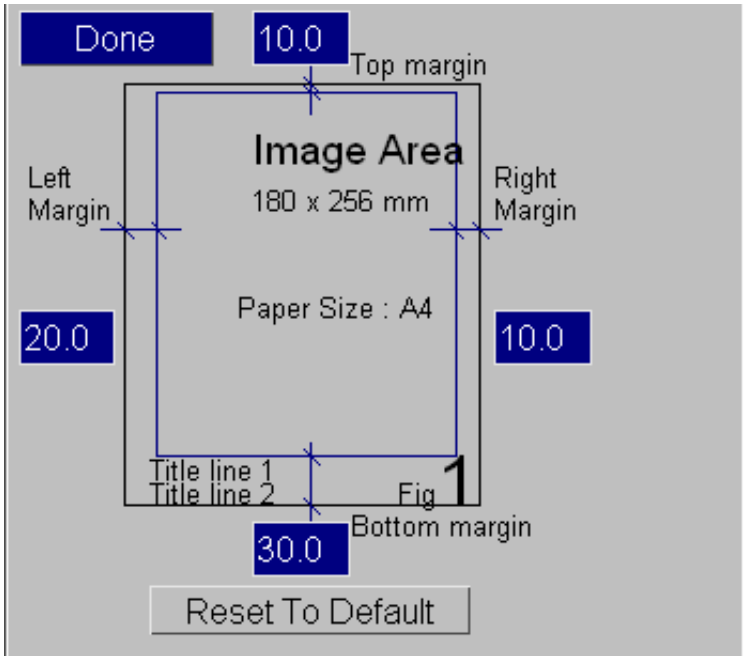
This flushes any remaining data to disk and disconnects the file from the programme.

Next section

7.3.2 **MARGINS...** Modifying laser paper size on the page.

The **MARGINS** button in the laser control panel gives a special sub-menu that allows you to select the margins on all sides:

The margins will only apply to the axis of the plot that comes closest to the paper borders; the other axis margins will be overridden to maintain the correct aspect ratio of plots (ie no image distortion).



7.3.3 Laser plotting during Animation.

It is possible to store the sequence of images generated for animation in laser files. (This was not possible in earlier releases.) The figures showing 3x3 Landscape and 2x4 Portrait multiple plots are examples of laser files created in this way.

To do this proceed as follows:

- Turn the laser switch on in the static part of the programme.
- Use **-> ANIMATE** to enter animation. You are warned that laser output is on, and have to confirm that it is genuinely required.
- Generate the animation sequence in the normal way. As each frame is created it is copied to laser the file just like an ordinary (static) plot.

The header on each frame is altered to show the animation frame number and corresponding time value. (Normally this would show the analysis title.)

The plot title is unchanged, but the figure number is replaced with the first frame number on each sheet.

Filenames are generated automatically if required using the standard naming sequence described in Section 7.1.2.

7.3.4 Creating Encapsulated Postscript (EPS) files.

EPS format is used by many software packages to import postscript images. The laser files written by D3PLOT are not in EPS format, but only two very simple edits at the top of the file are required to change this.

The first seven lines of any D3PLOT laser file look like this:

```
%!PS-Adobe-2.0
%%EndComments
%%Pages: 1
%%Page: 1 1
```

To convert it to EPS format you must add a "**%%BoundingBox:**" line, and delete the "**statusdict**" line. Thus this file becomes:

```
%!PS-Adobe-2.0
%%BoundingBox: 0 0 595 842
%%EndComments
%%Pages: 1
%%Page: 1 1
```

```
statusdict begin
                                /a1test save def
/a1test save def
```

The arguments of the "BoundingBox" line are the Postscript coordinates:

<lower left> <lower right> <upper left> <upper right>

These must be expressed in raw Postscript space of 72 points per inch, and they assume that the paper is in portrait format with its origin at its lower left corner.

The the values in the example above refer to A4 format: 210 x 297 mm = 595 x 842 points; US "letter" paper would give 8.5" x 11" = 612 x 792 points. Clearly a smaller bounding box would select only a subset of the image.

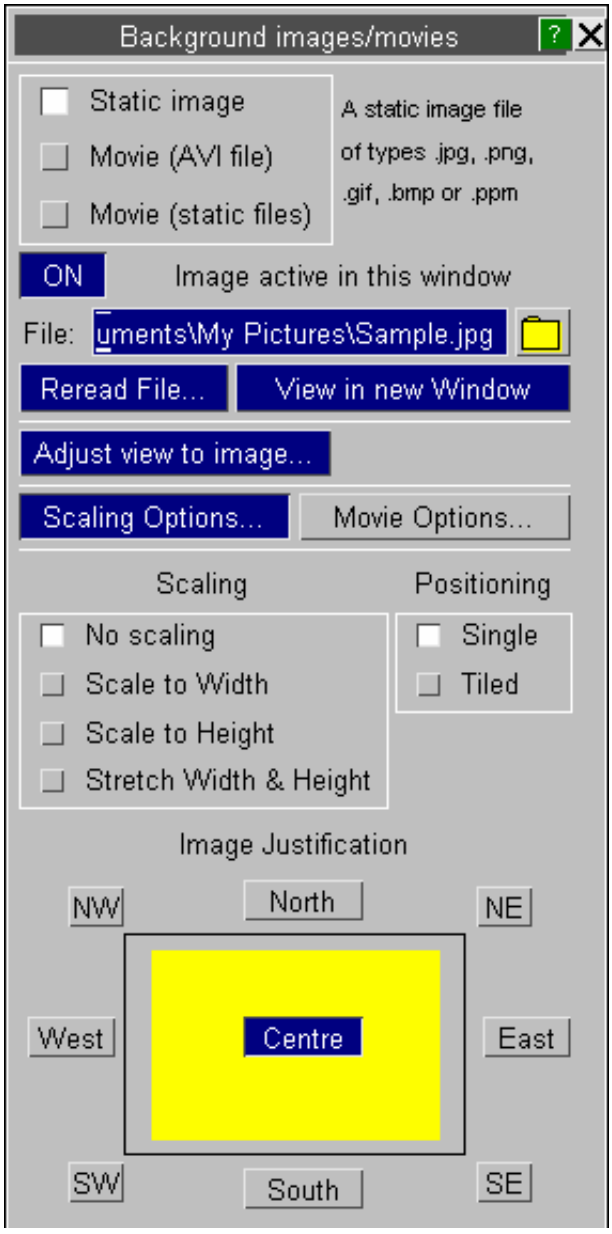
For more information on encapsulated postscript see the "PostScript Language Reference Manual, 2nd edition" by Adobe Systems Incorporated. (Published by Addison Wesley, ISBN 0-201-18127-4)

7.3.5 Notes on laser plotting

- Users on 3D devices should note that turning the laser on will temporarily force the graphics mode back to 2D. This is because a laser plot is intrinsically a 2D image and is computed in software.
- Transient graphics added "dynamically" to the screen are never copied to laser files. Examples are cursor-pick symbols, and also the information added interactively with the **DYNAMIC_LABEL** function.
- If an attempt to open a laser file fails because the file/directory refuses "write" permission, or the disk is full, you are warned and laser output is switched off.
- You can switch laser output **off** and **on** at will in the course of assembling a file with multiple images. Sub-images will only be written when the laser is on.
- Some of the defaults here may be preset outside D3PLOT via preferences in the **.oa_pref** file: see Appendix II

Next section

7.4 Reading static images and movies



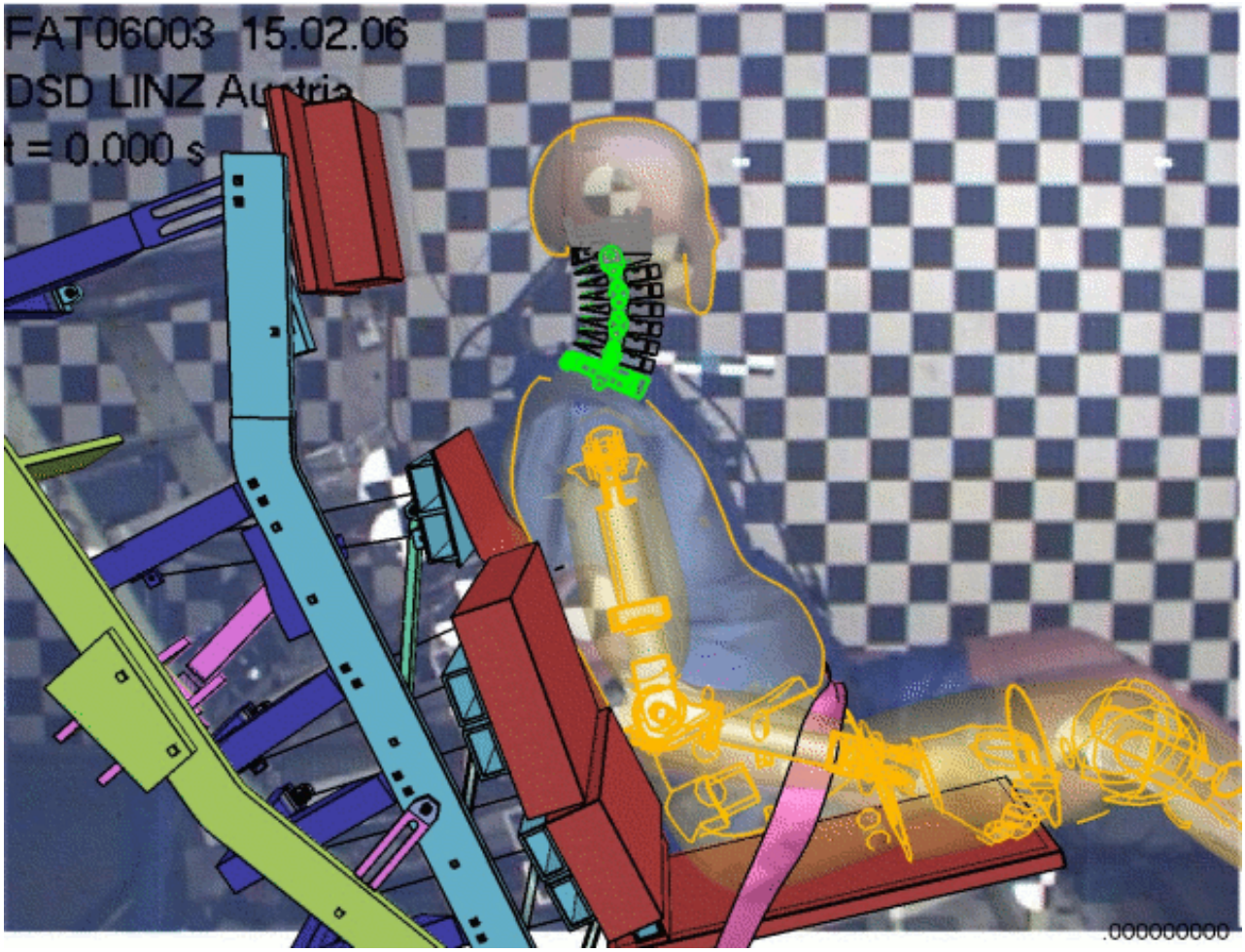
Static image
Read an image file to display as a background image behind a model instead of a solid background colour.

The formats we support are the same as we are able to write, see Capture.

Scaling Options

If the image dimensions do not match the graph window dimensions then the image can be scaled to fit or it can be tiled.

Below is an example background with the model overlayed on top.


FAT06003 15.02.06
DSD LINZ Au
t = 0.000 s
.000000000

MOVIE (AVI file)
Read movies in one of the following formats:

AVI `(.avi)`

MPEG `(.mpg)`

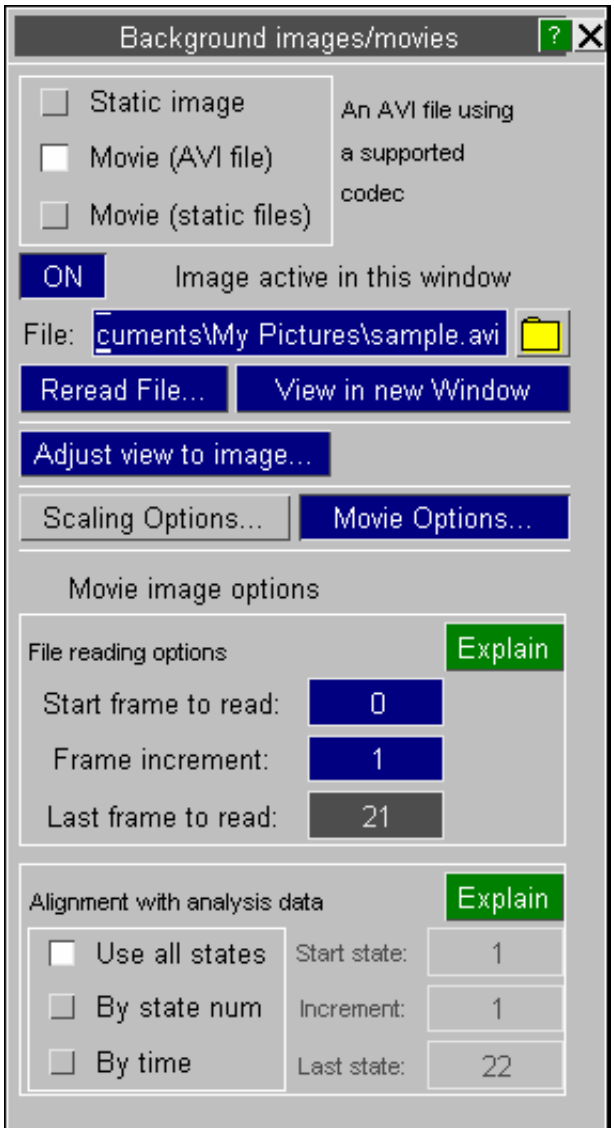
We support all movie formats that we are able to create, namely bitmap AVI, MJPG AVI and MPEG. Some AVIs need extra codec to be read in successfully, an error message will tell you that the codec is not supported and you will need to install the appropriate codec in order to read the movie.

Movie Options

Movie Options allow you to choose the start frame and state as well as the interval for both the movie and the simulation analysis. This can be particularly useful when you try to synchronize the movie with the simulation analysis. See Matching for how this can be done.

Scaling Options

If the movie dimensions do not match the graph window dimensions then the image can be scaled to fit or it can be tiled.



Movie (Static files)

Read a series of files with the same name and extension in a format of <name>nnn.<ext>

E.g.

d3plot001.jpg, d3plot002.jpg,
d3plot003.jpg.....d3plot010.jpg

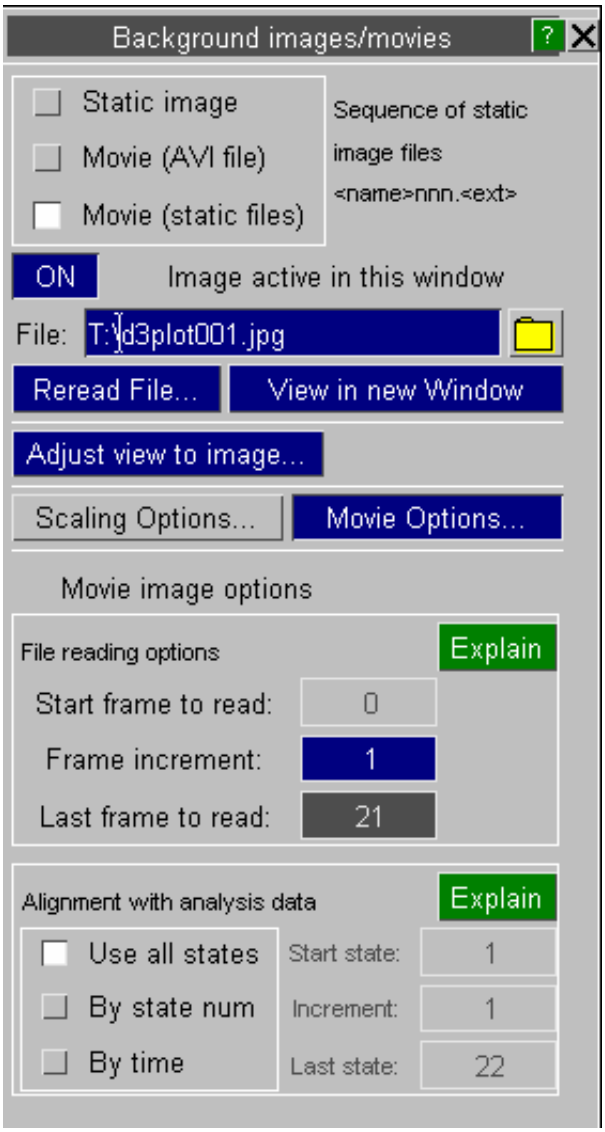
D3plot will search for all qualified images and read them together. D3plot will then display them in the order as they're numbered as you play the simulation.

Movie Options

You can control the start frame and interval the same way as you do with movie frames, and aligh them with analysis data, too. See Matching for how this can be done.

Scaling Options

If the image dimensions do not match the graph window dimensions then the image can be scaled to fit or it can be tiled. This will be applied to all images once you've set it.

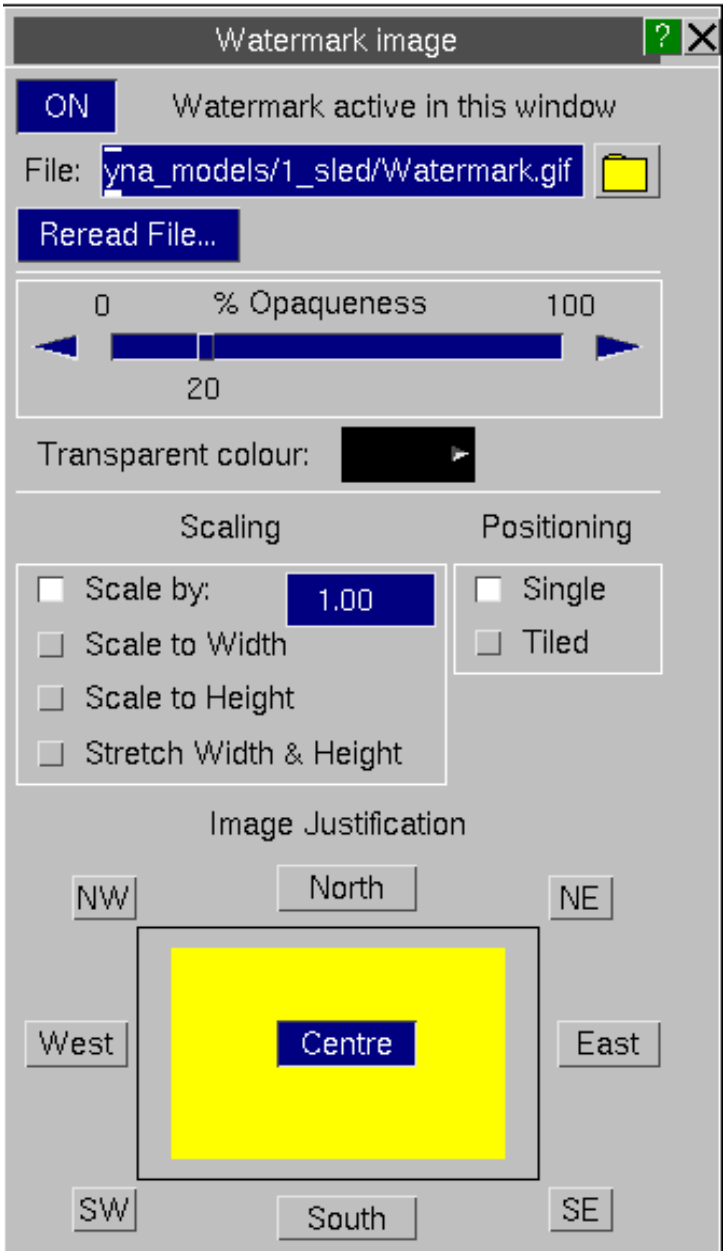


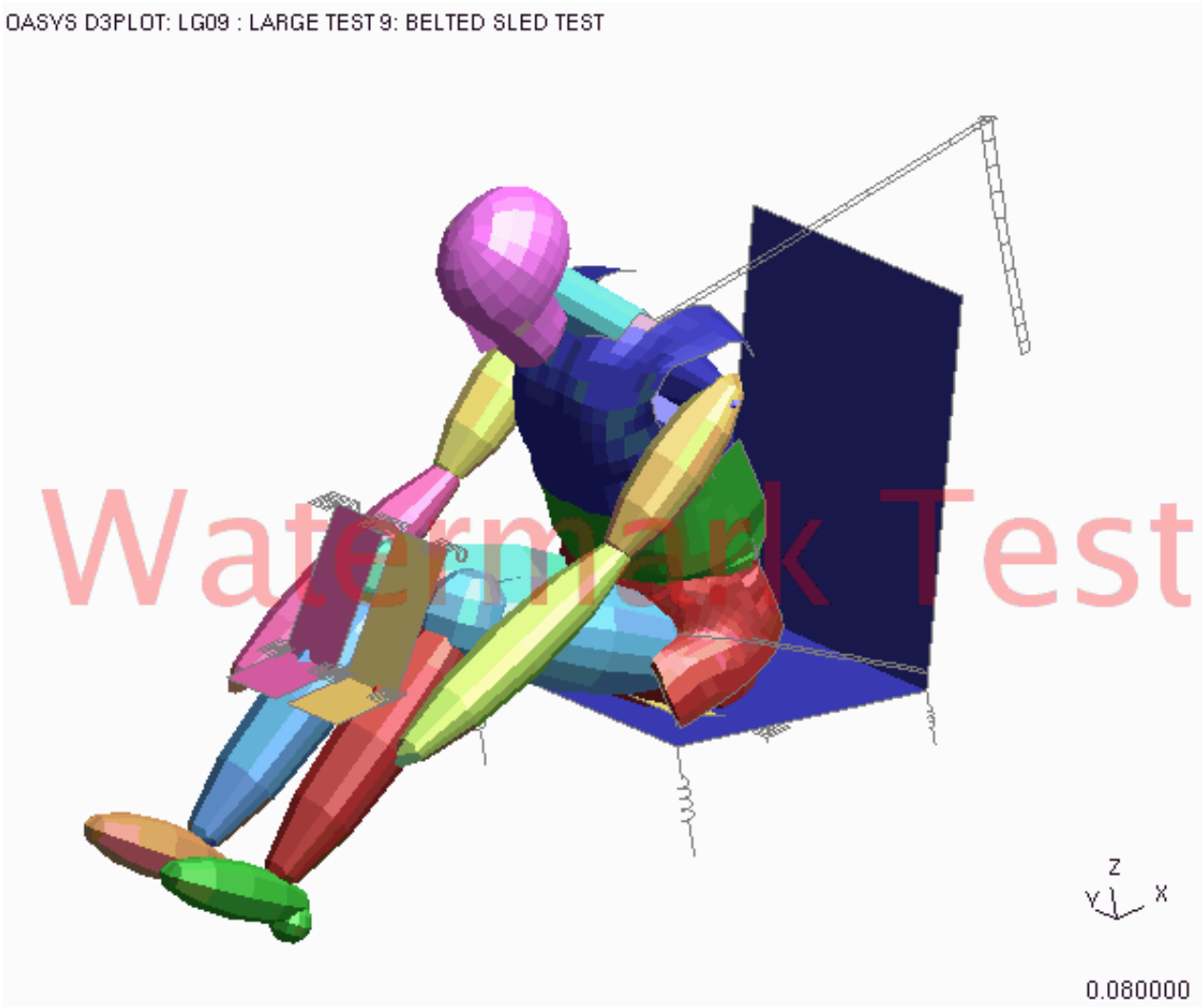
7.5 Watermarks

It is possible to add a "watermark" to a plot. Simply load in an image file in the watermark panel and set its transparent colour and overall transparency.

It will be drawn in front of the normal image, using the transparency settings you have defined. The position and size can also be set.

Below is an example with black as the transparent colour (the image was created on a black background) with 20% opaqueness.

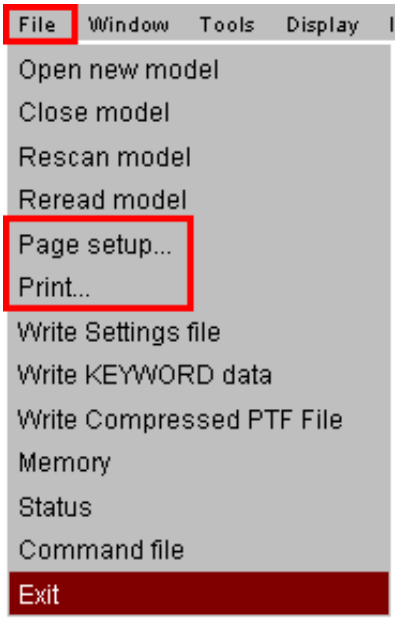




7.6 **Print** ... option (Windows platforms only)

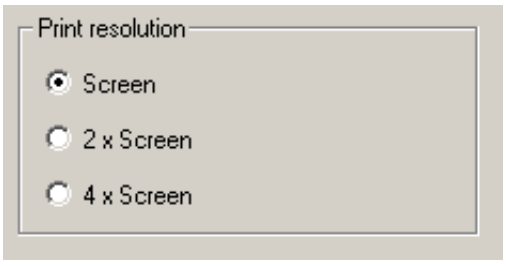
On Windows platforms only there are standard **Page setup** and **Print...** options under the **File** menu.

These will capture all currently visible graphics windows, size them to the current paper size rectangle and print them on the selected printer.



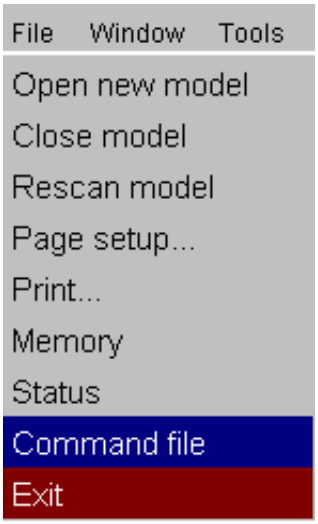
Print resolution:

On the standard **Print ...** panel there is an extra resolution option.



Choosing 2x or 4x resolution will capture the image at that factor times the current screen resolution, and will give a higher quality image. This may be useful if your screen is small, or you are planning to print on larger paper.

8 COMMAND AND SESSION FILES



8.0 Introduction to Command and Session Files.

It is possible to record commands issued during a D3PLOT session (in "session" files), and replay these during the current or a subsequent session (as "command" files).

This is exactly equivalent to having two tape-recorders available: one to record commands and one to play them back. It is also possible to have both turned on at the same time so that commands being played back from an earlier session get recorded (and possibly edited) during the current session.

Note that Settings Files may offer a more convenient alternative to command files, if the aim is to generate a particular plot.

8.0.1 How session files record screen-menu events

In command-line mode session files are simple: they simply record and play back verbatim typed in commands. When screen-menu mode is in use it becomes necessary to record commands expressed as button clicks, slider motions, etc; so a different strategy is required.

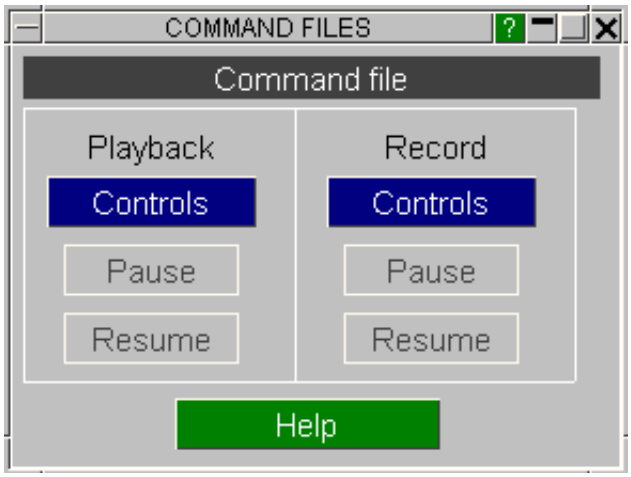
D3PLOT stores these screen menu "events" in special codes in the session file, and replays them as if they had been applied by the user.

8.0.2 Compatibility with session files from earlier releases of D3PLOT

Since command-line syntax has been preserved in this version, and command input into the dialogue window is still permitted, command-line command fiels are backwards c-compatible. Button-click command files are NOT backward-compatible.

The file format used for version 7.0 command files is itself backwards compatible: you should not need to modify existing files at all for them to work with this version.

8.1 **CFILE** Invoking the command-file launcher box.



Command and session files have separate control windows that are "launched" from the launcher box shown in the figure (right).

This can also be used during the operation of these files, see Section 8.4.

8.2 Recording “session” files.

This figure (right) shows the **RECORD** control panel for session files.

The **[RECORD] CONTROLS...** button in the launcher box maps this panel, which manages the recording of commands.

It works exactly like a tape-recorder: you can use the **RECORD** button to switch on recording, and all subsequent commands are recorded until it is turned off again by **STOP**.

It has two modes:

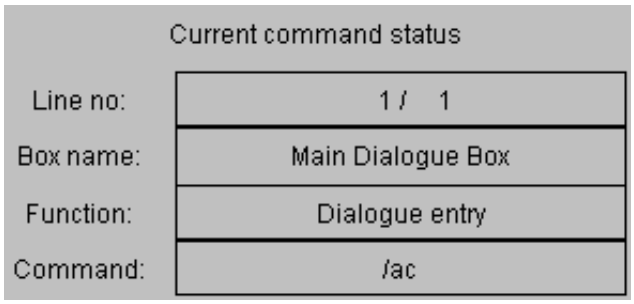
Record Actually records commands. This is the default.

Review Allows you to go back and forth to review recorded commands.

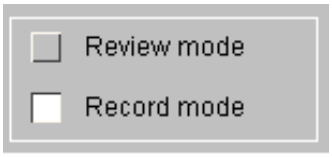
8.2.1 How “session” files are recorded.

When in **Record** mode, with the **RECORD** button depressed, every command, screen-pick, button press, etc (except those in the session & command file control panels) is recorded in an internal file. You could think of this as being the tape of a video recorder, with each video "frame" representing a command.

As each command is stored it is also reported in the **Current command status** area at the base of the box. Each command ha a unique line number, and in this example line #1 (out of 1 line recorded so far) is the command **`/ac`** in the dialogue box.



8.2.2 Reviewing stored commands.



If you switch from **Record** mode to **Review** mode then you will find that all the tape recorder buttons become live, and that (implicitly) command recording stops. You will also note that the **RECORD** button becomes a **REVIEW** button:

In review mode you can list stored commands using the commands shown here.

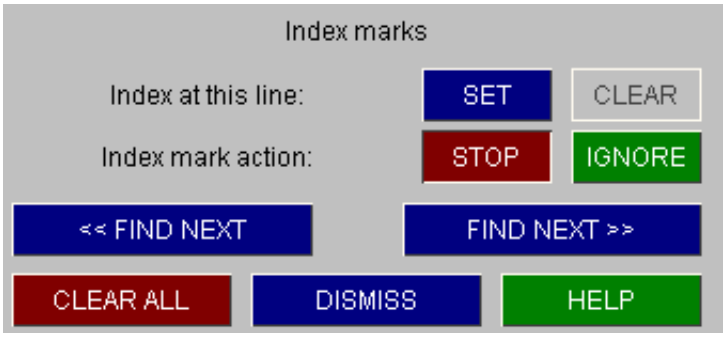
REVIEW > Plays the recorded commands forwards from the current position, listing them in the **Current command status** box as described above. A time delay of 1 second between commands is left by default, but you can alter this using the **SPEED...** options. **STOP** halts this process.

STEP < & > Respectively step backwards and backwards a single command.

SEARCH << & >> Respectively search backwards and forwards for a specified command.

You can search for any permutation of box name, function or dialogue string.

The options shown here are presented to you when you use either **SEARCH** button. It is recommended that you use the [**?**] buttons to identify boxes or functions from menus, as these will get the syntax right. Command string searches are not case sensitive.



TOP & END Will take you to line #1 and the last line respectively.

Goto line: Will take you to the line number you specify.

8.2.3 Editing and overwriting commands.

When you switch back into **Record** mode and start recording commands again, they start being recorded at the current position in the file. So if you have moved backwards during a **Review**, then start recording, you will overwrite commands stored at that point - just as would happen in an ordinary tape-recorder.

Commands are overwritten on a line-by-line basis. Consider this example:

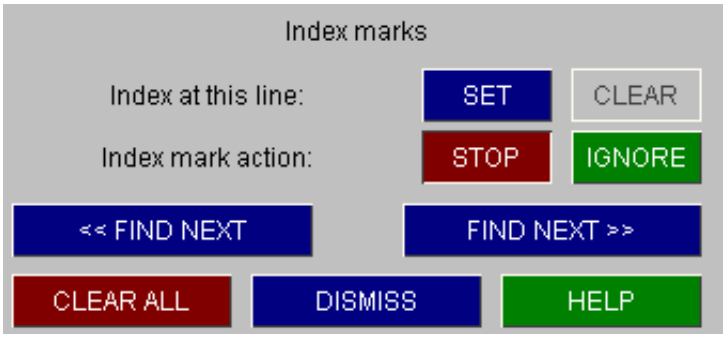
- You record 20 commands.
- You go back to line #13 in review mode.
- You record 4 commands starting at line #13.

In this situation the original four commands (#13 to #16) will be overwritten, but commands #17 to #20 will be unchanged.

To delete commands from the current position to end of file use the **DELETE TO EOF** button.

It is possible to edit command files using a text editor. WE recommend using command-line commands where these are available. These can be inserted in the file amongst button-click commands. To find out about the command-line menu system, type H or M in the dialogue box.

8.2.4 Inserting index marks.



You can put markers, "index marks", on any line in your file. These have no command significance, but are useful as targets for search operations. The **INDEX...** command options are shown here. Index marks can be **SET** and **CLEAR**ed, and you can search for them in either direction with the **FIND NEXT << & >>** buttons.

8.2.5 Writing out session files.

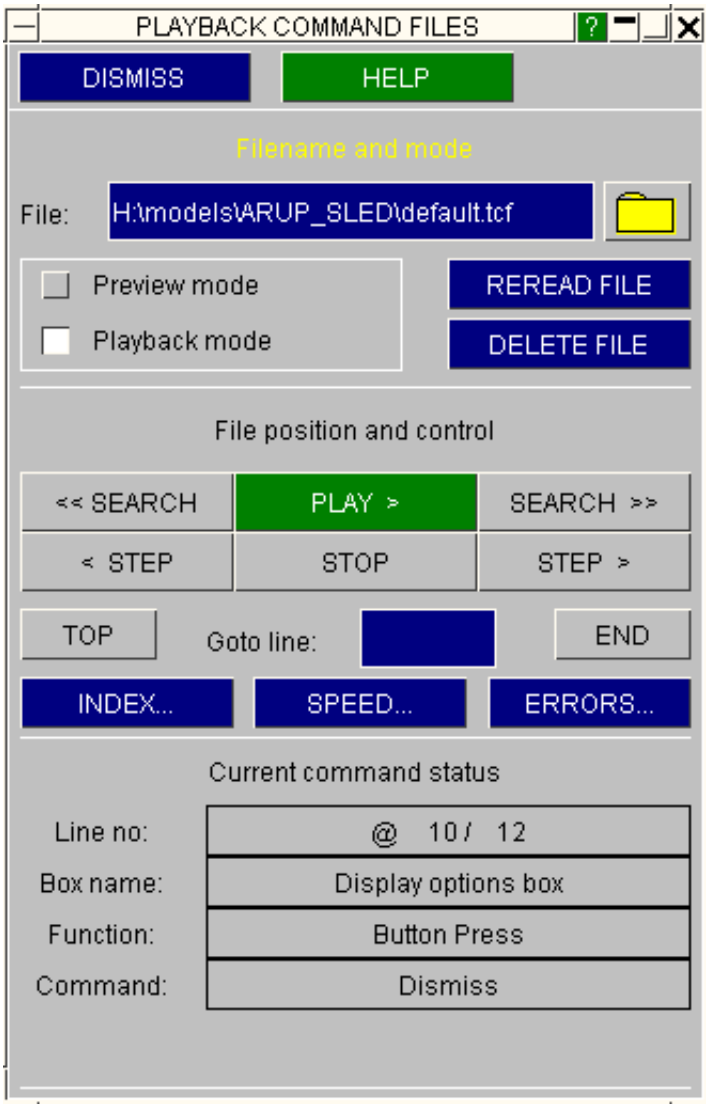
Session files are stored as internal scratch files, and must be written out to disk in ASCII form before they can be read back in as "command" files. This is done by the **SAVE TO DISK** command.

WARNING: If you record a session file, and then exit D3PLOT without issuing the SAVE_TO_DISK command, your recorded commands will be lost.

The format of the ASCII file, and advice on editing it, is given in Section 8.5.

Next section

8.3 Playing back “command” files.



This figure shows the command file playback control panel.

To use it you must first enter a command filename in the **File:** box. This is read in and converted into internal form, and the command buttons will then become live.

To execute commands continuously you use the **PLAY >** button. This can be halted with the **STOP** button. **STEP** executes commands one at a time.

It has two modes:

Playback Executes commands. This is the default.

Preview Previews commands without executing them.

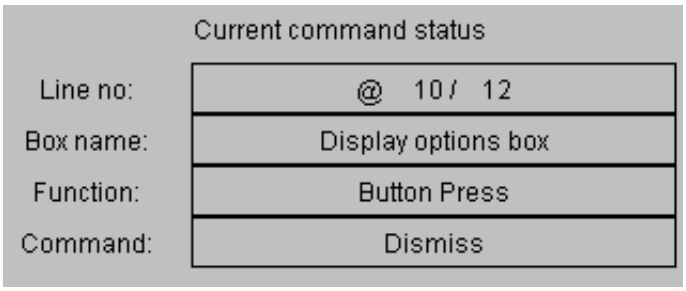
8.3.1 How “command” files are processed.

Commands are stored in an internal scratch file that, as with "record" mode, is like a tape in a tape recorder. Each command forms a separate line, and you can move the "tape" back and forth to locate it at any line.

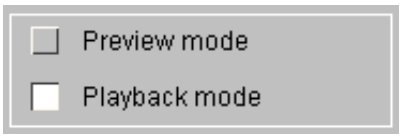
When in **Playback** mode you can **PLAY** your commands. Each command from the current position to <end of file> is executed as if you had typed it in, screen-picked it, etc. To stop a playback prematurely use the **STOP** button.

By default a **PLAY** operation operates at full speed, but you can introduce a delay between commands using the **SPEED...** options. This can be useful if you want to interrupt a sequence at a particular point, and if full speed playback is too fast to follow.

The current command is always shown in the **Current command status** box. In the example here we are currently at command #10 (out of 12 in the file), which is a **DISMISS** button press.



8.3.2 Previewing commands.



The playback control panel can be operated in **Preview** (instead of **Playback**) mode. This allows you to view commands without actually executing them, and thus to position yourself where you want in the file.

The **PLAY >** button is replaced by a **PREVIEW >** one to remind you which mode you are in. Commands are listed in the **Current command status** box as above, and you can **STEP** backwards and forwards as before. These and the other positioning and searching commands operate in exactly the same way as for recording session files: see Section 8.2.2 for more detailed instructions.

8.3.3 Handling errors during playback.

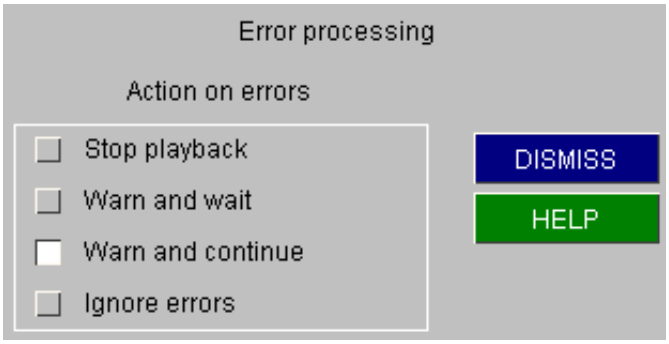
When you are playing commands it is possible to generate errors. Examples might be playing back commands recorded in a model with 10 materials in a different one with only five materials. If the screen button corresponding to the (non-existent) 10th material is picked in some context (eg from a menu) an error will occur.

The **ERRORS...** command lets you set the action to be taken when errors occur.

The **Action on errors** options are shown here, with the default **Warn and continue** option set.

The other options are self-explanatory.

Section 8.5 gives more information about how errors may occur, and how to avoid them.



8.3.4 Opening new command files.

You can read in a new command file at any time, or reread the current one (assuming it has been updated on disk).

In either case **all** existing commands in the internal scratch file are deleted and superseded by those read in. You cannot concatenate the contents of the new file with the existing one.

8.3.5 Recording session files during command file playback.

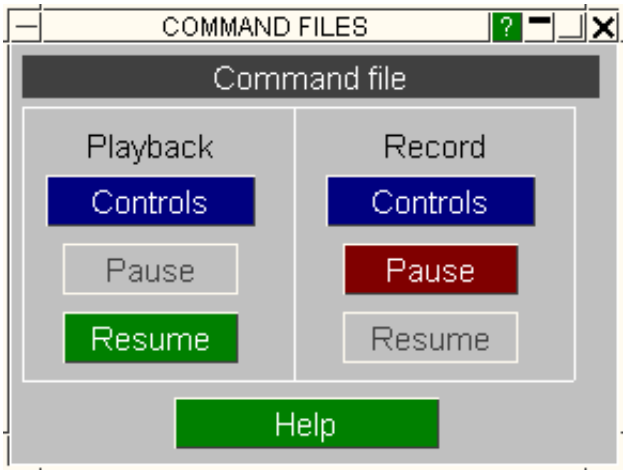
This is quite legal. The two operations are quite separate, and commands executed from a command file will be recorded exactly as if they had been typed in. This is the recommended method of editing and appending to session files.

8.4 Using the “launcher” box during recording and playback

This figure shows the command/session file "launcher" box part of the way through a simultaneous record and playback session.

This box need not be mapped, but it can be useful to use it rather than the full control panels since it uses less space.

The "launcher" box always echos the **PLAYBACK/STOP** status of the playback box if a command file is resident in memory.



It also echos the **RECORD/STOP** status of the record box if that is in use.

Since the main panels for these two operations take up a lot of space on the screen you can use the simple on/off functionality provided by the "launcher" box instead.

Think of it as a primitive remote control that can only has a "pause" function. It operates as follows:

During Playback:	When you are in **PLAY** mode the **PAUSE** button will be available, and you can press it to pause playback. Once pressed, as in the example above, it will be greyed out and the **RESUME** button made available instead. This, obviously, continues the playback.
During Recording:	When **RECORD** is on, as in the example here, the **PAUSE** button is live. If pushed it suspends recording, and subsequent commands will not be recorded until **RESUME** is pressed.

In either case you can of course invoke the main control panels and override these buttons.

When recording or playback are not active then the buttons on this panel will be greyed out: they only become "live" when they might be needed.

8.5 More information about command and session files

8.5.1 The format of the ASCII disk files used for these.

The ASCII files are up to 132 columns wide, with column numbers reserved as follows:

1 to 80	Dialogue strings from the dialogue box, button text, etc
81	Continuation character for long strings (**C** below)
82 to 132	Encoded screen-menu functions

Diagrammatically this gives:

```
<Dialogue string (80 characters)> |C| <Encoded screen-menu function values>
```

If the continuation column (#81) is occupied then it assumed to be a "long" line, and the character string can extend to column 132. Any screen-menu function values will then appear on the next line.

If columns 82 to 132 are empty the command is assumed to be a simple "dialogue" command, typed in by the user.

An example of a command file is:

```
$
$ Oasys D3PLOT version 7.0 session file
$ Date/time: 25-Nov-95 17:42:00
$
/re 2                                   1    1    1    0    0    0    0    0
/bl sta                                 1    1    1    0    0    0    0    0
/hi                                     1    1    1    0    0    0    0    0
DISMISS                                26    3    2    1    0    0    0    0
CFILE                                  23    3    2   14    0    0    0    0
```

8.5.2 Backwards compatibility with command files from earlier versions of D3PLOT

This file format is fully backwards compatible with text-only command files from earlier versions of D3PLOT. They will have nothing in columns 82 to 132, and so will be treated as command-line input, which is what they are.

The command-line syntax in Version 7.0 of D3PLOT is virtually unchanged from earlier releases, so the vast majority of existing command-files should run with no modification.

8.5.3 Editing command files by hand.

You can type in commands (in columns 1 to 80) that are genuine "command-line" commands, and you should leave columns 82 to 132 blank.

It is also possible to concatenate screen-menu command files using the editor, **BUT** beware the following:

Context errors: The instructions "drive along motorway", and "remove wheel nuts and take wheel off car" are both perfectly sensible. But only if the intervening commands "in case of puncture pull over to hard shoulder, stop, and jack up car" are inserted.

If you concatenate two sets of command files you can generate context errors if you miss out intervening steps. For example if file one sets up contour levels, and file two operates animation controls, you will have problems because you have omitted to switch from "static" to "animation" modes in between.

This is a context error. Normally you would not be able to operate animation controls while in "static" mode, since the buttons would not be available. But this protection is absent when replaying command files.

Range errors: Menus of items, for example lists of materials, may have fewer rows in this file than in the one where the command file was generated.

Since the screen-menu function recorded is the menu row, not its contents, this produces errors if you try to reference rows for non-existent entries.

To concatenate files it is better to **PLAY** each of them in turn through D3PLOT, while simultaneously **RECORD**ing them. This will allow you to spot and fix context errors more quickly, and also to insert missing commands if needed.

8.5.4 Certain commands are not recorded in session files.

Commands in "pop-up" windows, for example warning, confirmation and listing boxes are not recorded in session files.

Also commands in the file filter box are not. You can make this work for you: if you want to define a different filename at playback time use [**?**] for the file filter box, if you want to use a fixed filename without intervention at playback time type it explicitly into the **File:** text box as this will be recorded.

8.6 Associating command files with Function keys

From version 8.3 onwards the keyboard function keys **F1** to **F12** can be programmed to play back command files. This is described under **UTILITIES, FUNCTION KEYS** in section 6.9.11.

8.7 Running command files from the command line.

When running D3PLOT in batch it can be convenient to invoke a command file to run a specified set of commands. The following command-line arguments may be used to do this:

-cf=<*command filename*>	Runs "command filename" until its end, and then reverts to normal command line input.
-exit	This optional. If used then D3PLOT exits when the end of the command file is reached.

More information about command-line syntax is given in Appendix IV.

9 DISPLAY_OPTIONS:

Controlling the appearance of the display and many of the items drawn on it.

The main **Display Options** control panel is shown right.

This contains switches (left column), and options with sub-menus (right column), which control many aspects of display appearance.

See the following sections for details:

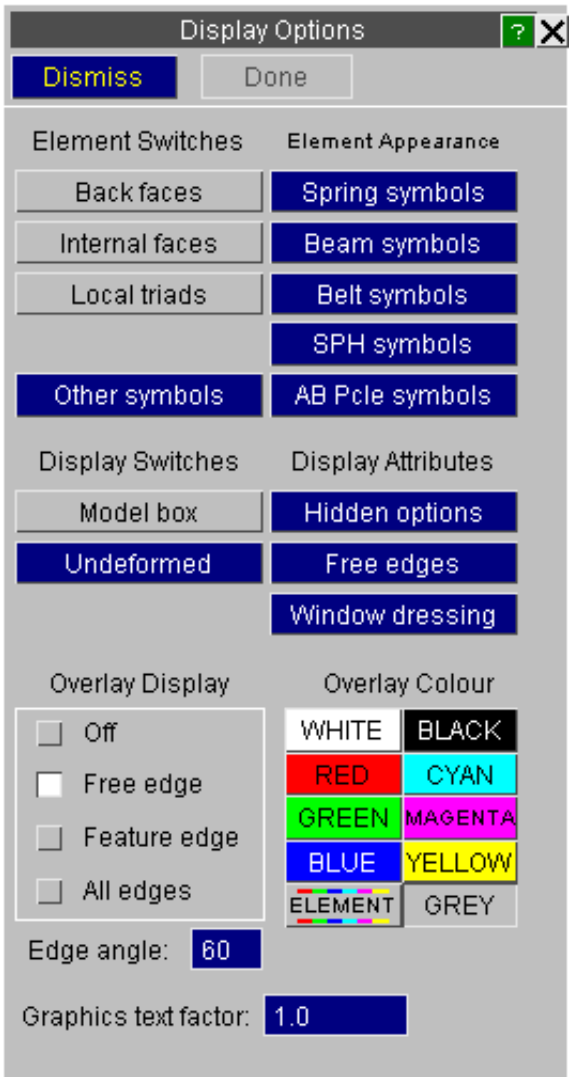
9.1 Back faces
9.2 Internal faces
9.3 Local Triads
9.4 Model box
9.5 Undeformed

9.6 Spring symbols
9.7 Beam symbols
9.8 Belt symbols
9.9 SPH symbols
9.10 Airbag Particle symbols
9.11 Other symbols

9.12 Hidden options
9.13 Free edges
9.14 Window dressing

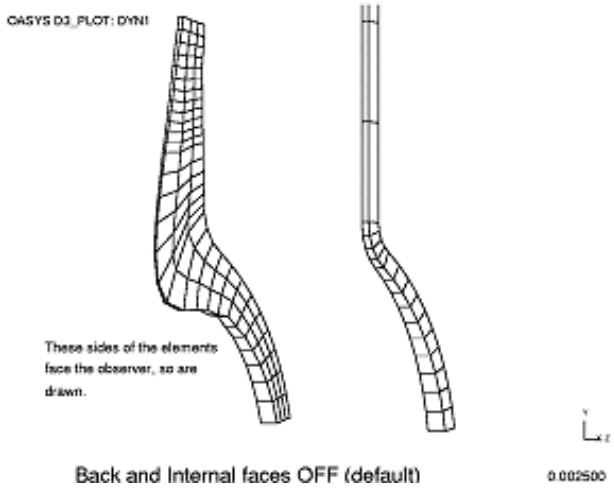
Overlay display, colour and edge angle are described in section 4.2.2.2

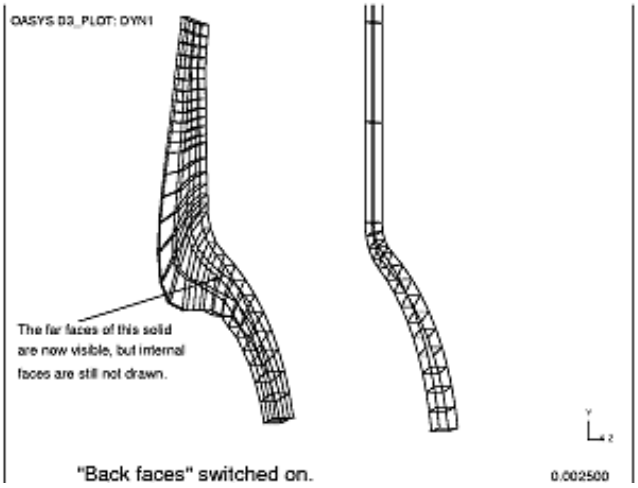
9.15 Graphics test factor



9.1 **BACK_FACES** switch: Display of “back” faces of solid and thick shell elements

To save graphics rendering effort the "back" faces of 3D elements (solids and thick shells) can be turned off. This is the default on 2D devices, but on 3D devices they are left on (in case the model is rotated to expose its back). The effect of this is shown in the figures below.

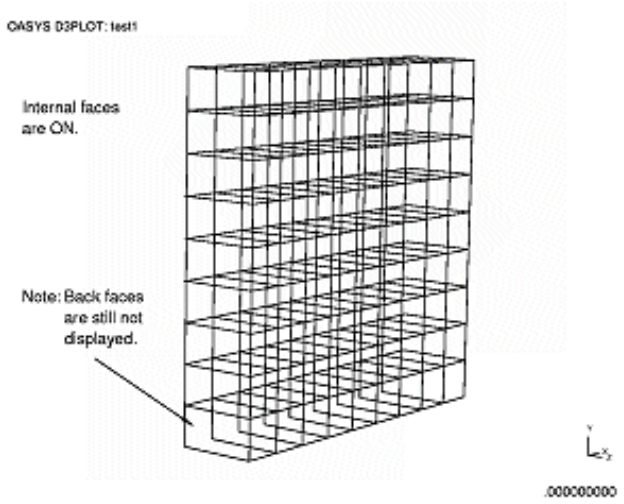

OASYS D3_PLOT: DYN1
These sides of the elements face the observer, so are drawn.
Back and Internal faces OFF (default)
0.002500

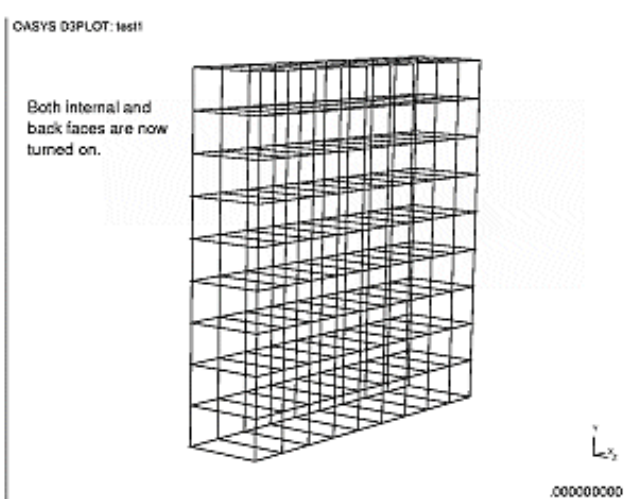

OASYS D3_PLOT: DYN1
The far faces of this solid are now visible, but internal faces are still not drawn.
"Back faces" switched on.
0.002500

BACK_FACES off **BACK_FACES** on

9.2 **INTERNAL_FACES** switch: Display of inside faces of solid & thick shell elements

By default the internal faces of 3D elements are never displayed: as well as requiring extra rendering effort leads to cluttered plots. The figure below left shows a wall of solids with internal (but not back) faces turned on, and below right, the same wall with both internal and back faces on.

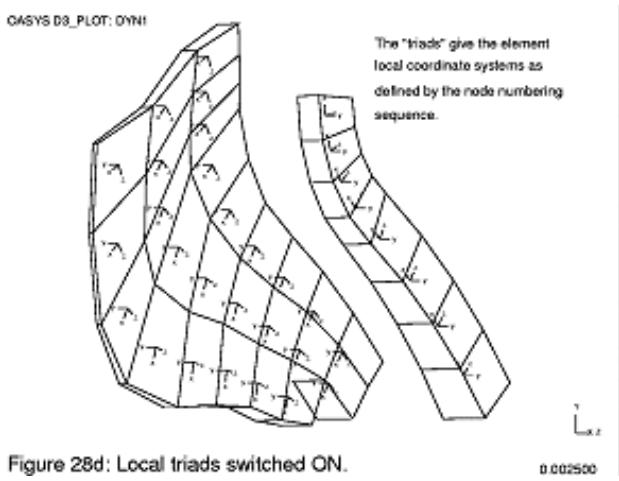




You may need to turn internal faces on if you are trying to screen pick solids several layers deep by screen-area. Otherwise each screen area selection will only pick the (visible) solids at the front of the image, like peeling an onion, and several such picks will be required to select all the solids through the depth.

9.3 **LOCAL_TRIADS** switch: Display of element local axes

Solid, shell, thick-shell, beam and interface elements all have local coordinate systems, and you can display "local" triads of their axes using this switch. The figure below left shows examples of local triads on solids, and the figure below right, on beams and shells.


OASYS D3_PLOT: OYN1
The "triads" give the element local coordinate systems as defined by the node numbering sequence.
0.002500

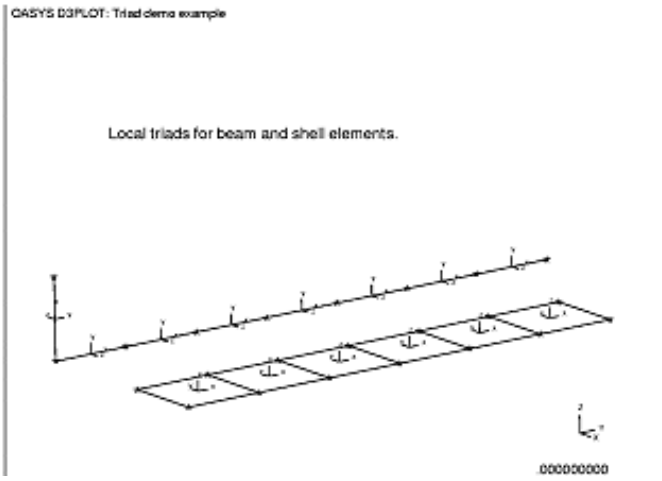


Figure 28d: Local triads switched ON.

The method used to compute element local axes is given in the following section:

Solids	Section 12.7.4	In all cases D3PLOT generates local axes from the element topology alone. ***Note that it does NOT currently "know" about any local material axes, beta angles or ply directions.***
Thin shells	Section 12.8.2.1	
Thick shells	Section 12.9.6	
Beams	Section 12.10.1	
Segments	Contact segments use the same method as thin shells.	

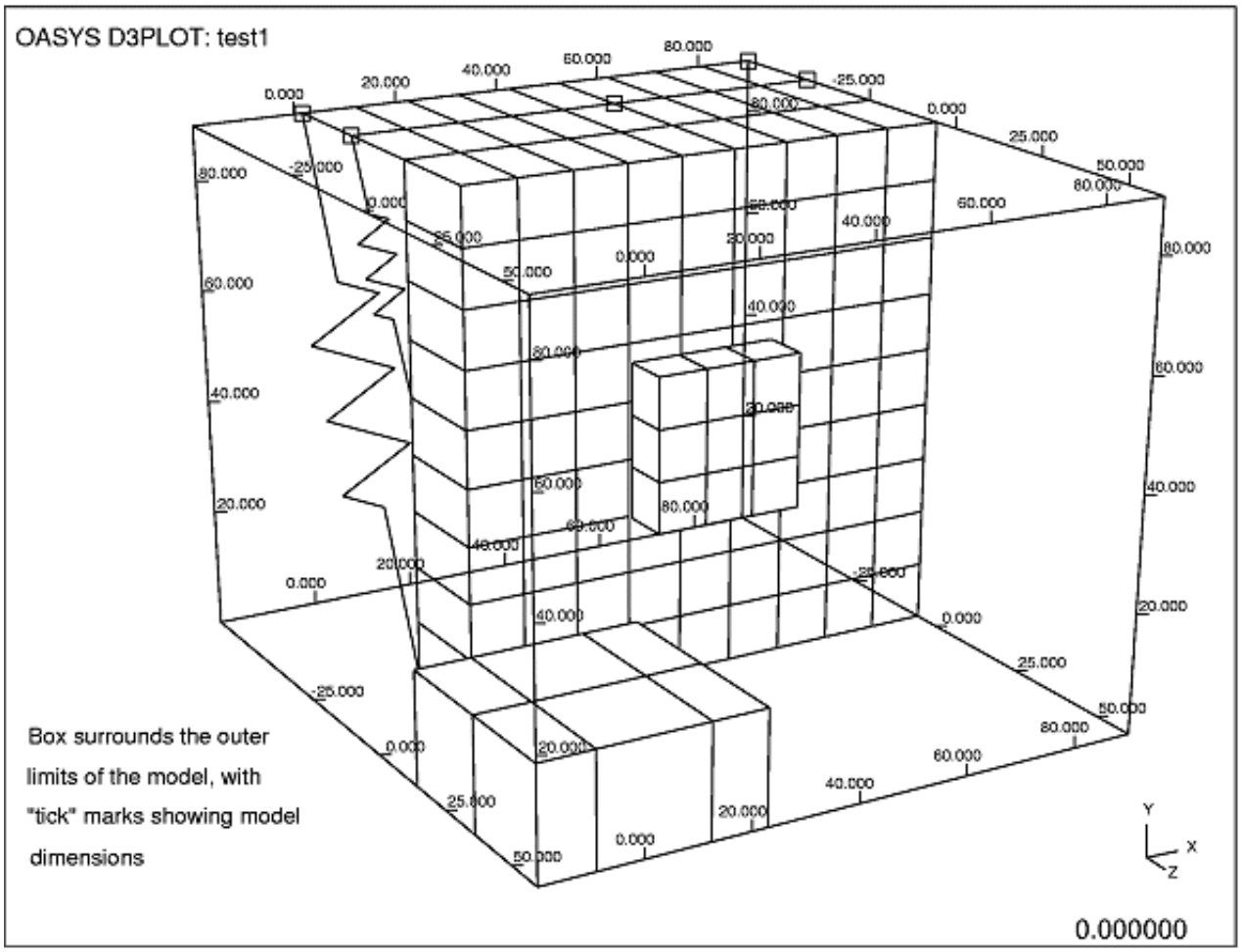
An easier way of spotting reversed outward normals of shells and interface segments

The local Z axes (outward normals) of shells and interface segments are important in some contexts. For shells averaging across adjacent elements with top and bottom surfaces flipped gives misleading contours, and for interface segments some types of contact surface cause problems if their outward normals face the wrong way.

Therefore, as well as being able to display local triads, you can plot their **OUTWARD_NORMAL** components in **CT** continuous-tone plots. This plots normals that face you in magenta, and those that face away from you on blue. It makes it easy to spot elements that face the wrong way: certainly it is easier than hunting for reversed local Z axes using triads!

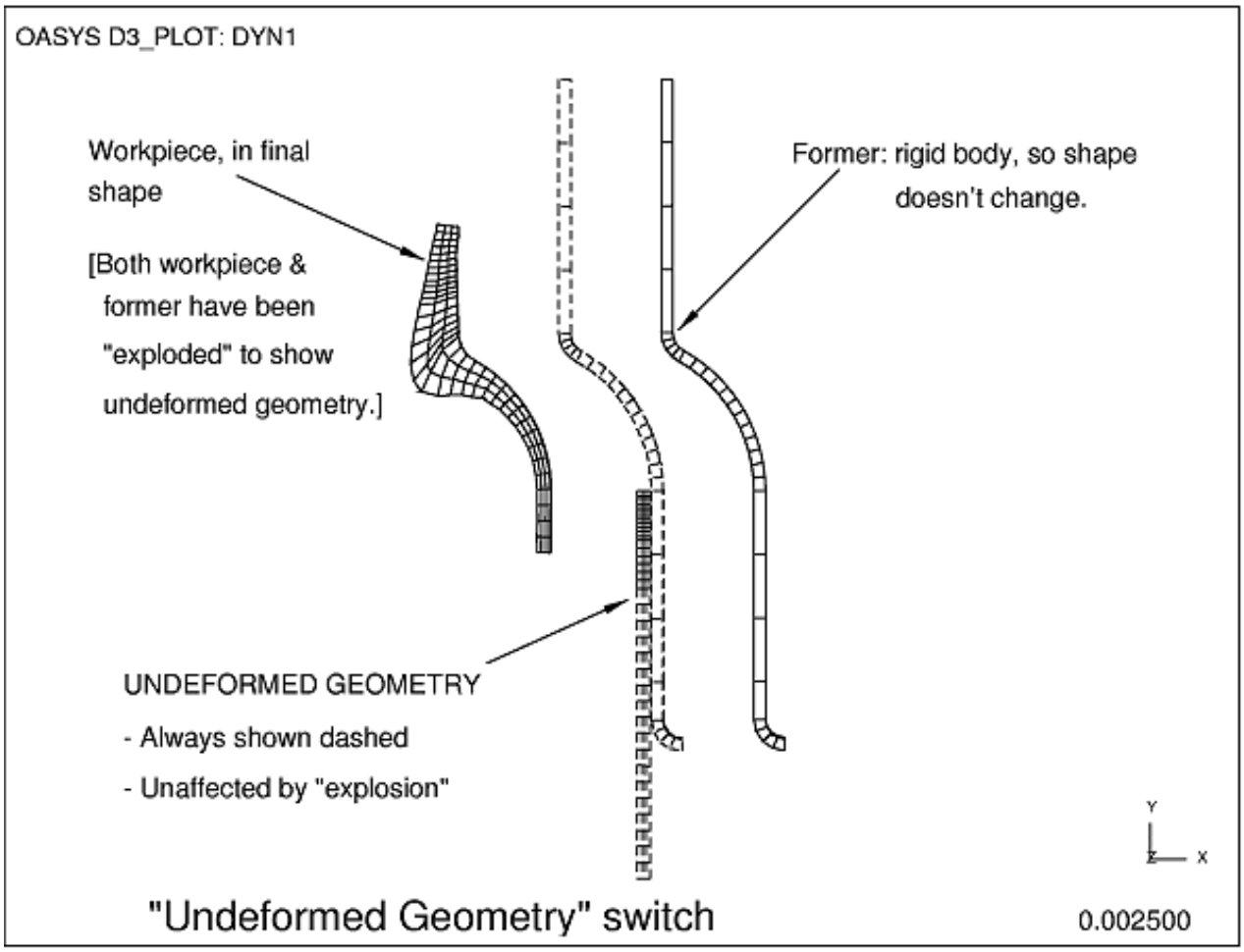
9.4 **MODEL_BOX** switch: Displaying the model external dimensions

It can be useful to know what the external dimensions of your model are. One way to do this is to use a **GRATICULE** (see Section 9.14), but this uses screen space and can't be transformed with the model. The **MODEL_BOX** option is an alternative: it draws a box that represents exactly the largest [x,y,z] external dimensions of your model, and adds tick marks. The figure below shows an example of this. This box rotates with the model, and can be viewed from any angle.



9.5 **UNDEFORMED...** menu Displaying the undeformed geometry

Normally just the current geometry is drawn, but you can overlay this with a hidden-line plot, using in various styles, of the undeformed geometry. An example is shown below.



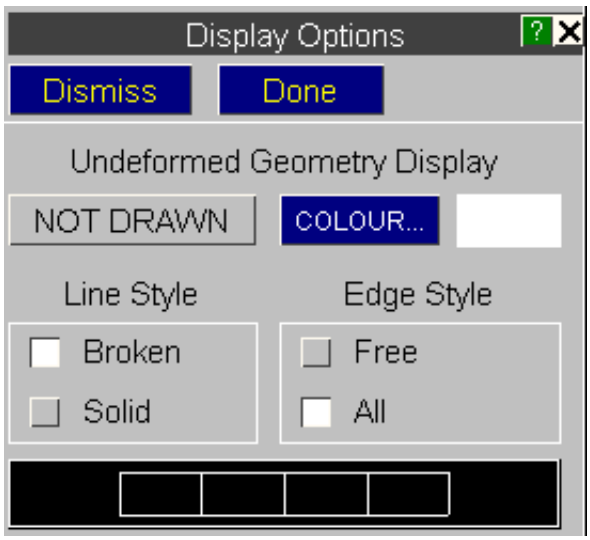
The following attributes of the undeformed geometry may be set:

(NOT_)DRAWN Whether it is displayed.

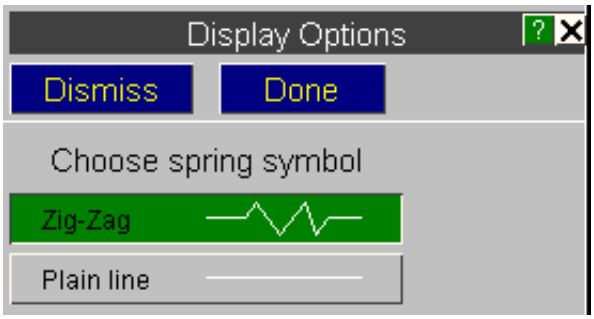
COLOUR Its line colour

Line Style May be **Broken** or **Solid**

Edge Style May be **All** edges or **Free** edges only.

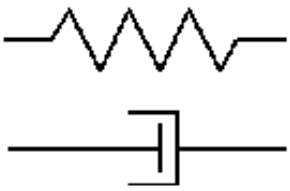


9.6 **SPRING_SYMBOLS...** menu: Setting the drawing style for springs and dampers



By default springs use "zig-zags"

and damper elements use "dashpots"



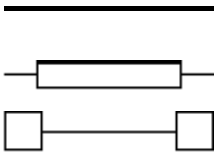
You can change these to **Plain line** style, in which they are drawn as simple lines. This saves vectors, reduces clutter, and can be useful during animation to improve speed and economise on memory usage. See Section 9.9 for details of how grounded and zero length elements are drawn

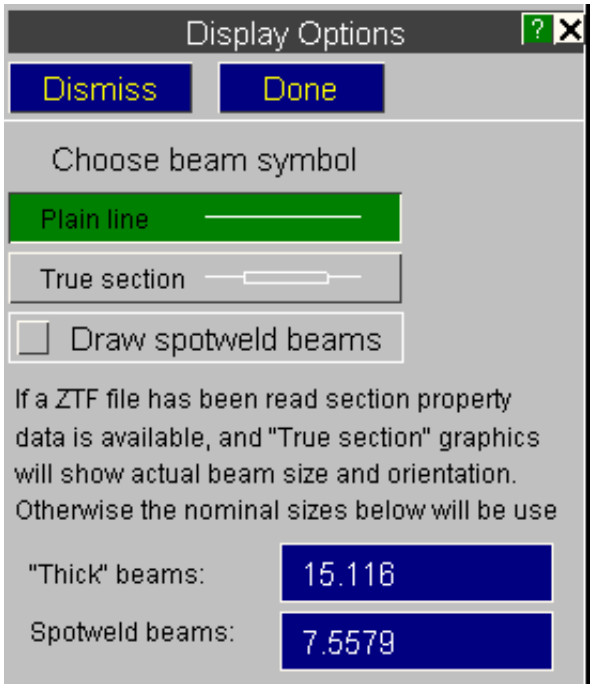
9.7 **BEAM_SYMBOLS...** menu: Setting the drawing style for beams.

By default beams are drawn as lines

You can choose to use "thick" lines

"Thick" and spotweld beam symbol sizes may be set individually.




Display Options
Dismiss
Done
Choose beam symbol
Plain line
True section
Draw spotweld beams
If a ZTF file has been read section property data is available, and "True section" graphics will show actual beam size and orientation. Otherwise the nominal sizes below will be use
"Thick" beams:
15.116
Spotweld beams:
7.5579

The "thick" symbols can make it easier to see beams in plots that contain a lot of lines, although they are slower to draw. The nominal dimensions in the boxes below are calculated as a %age of the overall model diagonal length, and they may require adjustment to give sensibly sized symbols for very asymmetrical models.

Similarly the size of the "blobs" on the end of spotweld beams may need adjusting to give visually acceptable results.

Note that "spotweld" (type 9 referencing ***MAT_SPOTWELD**) beams will only be drawn as such if a **ZTF** file for this analysis is found, as this contains the extra section data required to determine their attributes.

9.7.1 **True Section**: displaying the actual beam cross-section.

If a .ZTF file from release 9.2 or later of Oasys PRIMER has been read then section information will be available for beams which, if this option is selected, allows them to be drawn using their "true" section shapes.

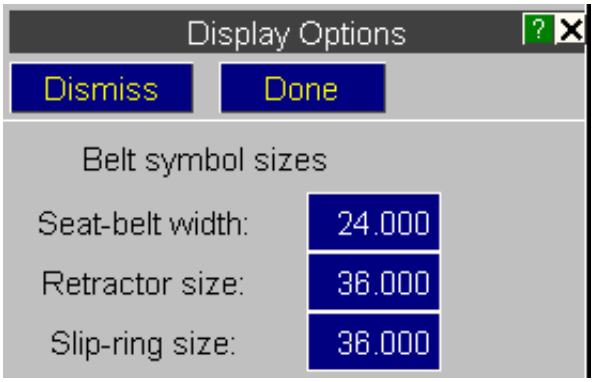
For beams with explicit section dimensions this is simple, and the actual dimensions are used.

For beams where only Area, Ixx and Iyy properties are available then a thin-walled rectangular section that matches these properties is synthesised. This should be approximately correct, but obviously it cannot represent I beams or rectangular sections with varying wall thicknesses, but it should give a reasonable representation of beam dimensions. If you use inconsistent or impossible properties you may get some strange looking sections!

9.8 **BELT_SYMBOLS...** menu: Setting the sizes of seat-belt and related symbols.

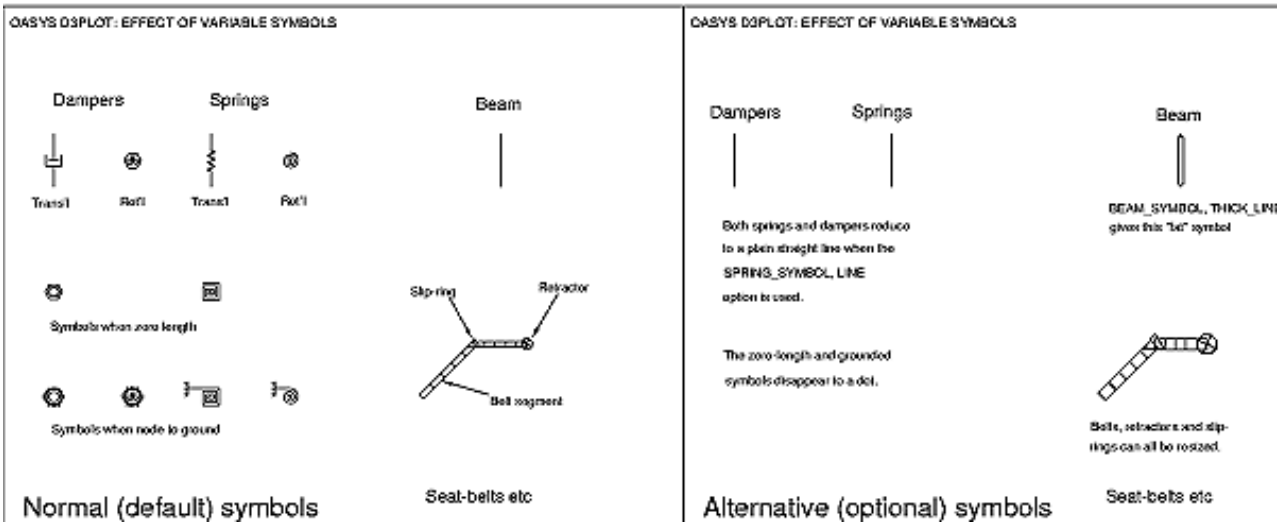
You cannot change the symbols used for drawing seat-belts and related elements, but you can change their visual size in this menu.

All dimensions are expressed in model space units and can be changed at will, the default values are only an estimate of what should look sensible.



Summary of default and modifiable symbols of Springs, Beams and Seatbelts

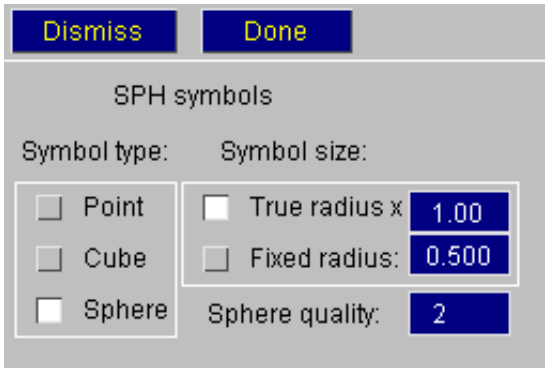
The figure below left shows default symbols for all springs, beams and belt types; and the figure below right shows what they can be modified to become.



9.9 SPH Symbols. Managing SPH element display.

Smooth Particle Hydrodynamic (SPH) elements are spheres with a given "radius of influence" that can change over time.

By default D3PLOT renders them as spheres of the current diameter, treated as opaque for shading and hidden surface removal, but this can be changed using the options here.



Symbol type: controlling the method used to draw SPH elements:

Point Uses "points" to display elements. These are 2 dimensional squares drawn in the plane of the screen at the appropriate location, and with width and height "radius" x 2.

OpenGL renders "points" extremely fast, meaning that this is an efficient display method if you have many SPH elements; however the symbol has no depth, is not "lit", and there will be a hardware-specific maximum size for a point symbol so it may not show the true size of elements.

Cube Uses a cube of width, height and depth "radius" x 2 to display elements.

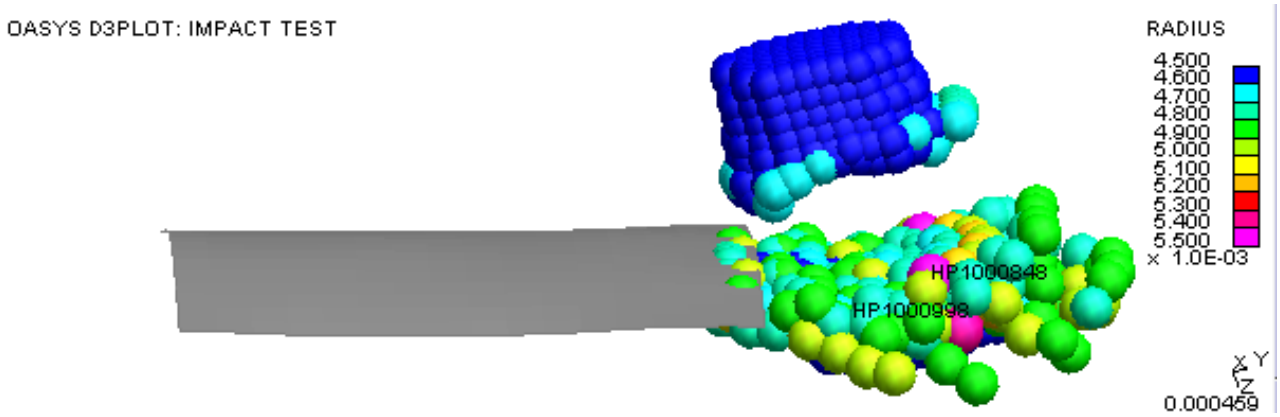
This only ever shows 3 faces, so it is reasonably fast to draw, but it looks a bit odd showing a spherical element as a cube. However cubes have depth and orientation, and can be lit, so the result is better-looking that a "point".

Sphere Draws a sphere of the relevant radius, giving a "true" element appearance. However spheres require many facets for rendering, making these slower to draw.

Sphere quality is a value between 1 and 5 which determines the number of facets used to render the sphere symbols. Each increment halves the equatorial and meridional angular increment size, quadrupling the number of facets on the symbol.

The default value of 2 gives a good compromise between symbol quality and rendering speed, but you may wish to increase this figure when generating presentation quality images.

This image shows a "chicken" made of SPH elements striking a flat plate, and demonstrates the default spherical symbols at quality level 2 and their true radius. The SPH elements have been contoured with their radius.



Symbol size: controlling SPH symbol display.

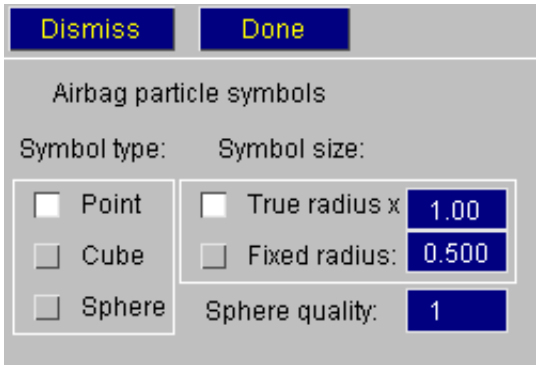
True radius x <*factor*> uses the radius of influence reported for the current state multipled by <factor>. This is the default and is recommended for most cases.

Fixed radius <*value*> uses the stipulated fixed radius size. This can sometimes be useful if analyses have gone wrong somehow and delivered very large (or small) radii, making it impossible to see what is happening unless the symbol size is set explicitly.

9.10 **AB Pcle Symbols**: Managing Airbag Particle display

Airbag particles can be thought of as small spheres of gas emitted from an inflator to apply pressure inside an airbag. D3PLOT treats them as "elements" for display and contouring purposes.

They are rendered using symbols that are the same as those used for SPH elements. However the two element types are highly unlikely to appear in the same analysis, and their relative sizes are also likely to be very different, so despite the potential for confusion this similarity is not a problem.



Symbol type: controlling the method used to draw ABP elements:

Point Uses "points" to display particles. These are 2 dimensional squares drawn in the plane of the screen at the appropriate location, and with width and height "radius" x 2.

"Points" are drawn extremely fast in OpenGL, and since a typical airbag may have tens of thousands of particles this is the default display method.

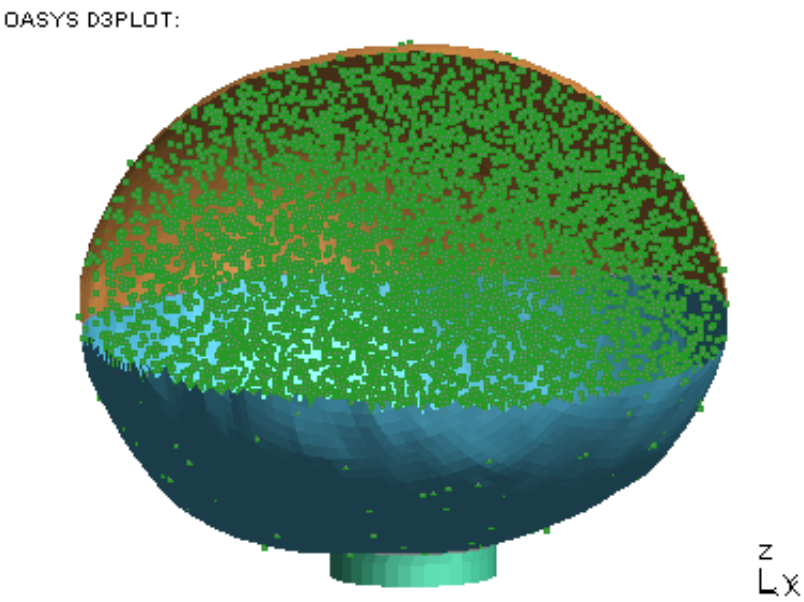
Cube Uses a cube of width, height and depth "radius" x 2 to display particles.

This only ever shows 3 faces, so it is reasonably fast to draw, but it looks a bit odd showing a spherical element as a cube. However cubes have depth and orientation, and can be lit, so the result is better-looking that a "point".

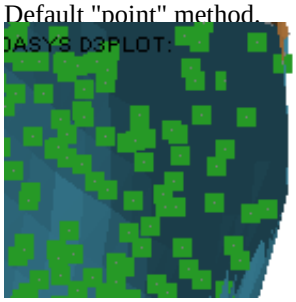
Sphere Draws a sphere of the relevant radius, giving a "true" particle appearance. However spheres require many facets for rendering, making these slower to draw.

Sphere quality is a value between 1 and 5 which determines the number of facets used to render the sphere symbols. Each increment halves the equatorial and meridional angular increment size, quadrupling the number of facets on the symbol.

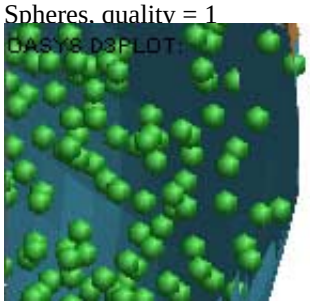
The default value of 1 gives a rather "pointy" looking particle symbol, but it is usually acceptable given their small size. Higher quality values may be necessary when generating images for presentation.



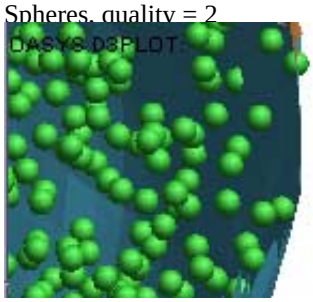
Default "point" method.



Spheres, quality = 1



Spheres, quality = 2



This image shows a typical "pancake" airbag being inflated by particles. Part of the fabric has been cut away to show the particle elements inside which are drawn by the default "point" method.

The three images on the right show a detail of the right hand side to demonstrate how the different display methods appear when enlarged. For this particular analysis the "point" display method the speed of animation was limited by the refresh rate of the monitor at 85 frames per second, whereas the "sphere" methods displayed at about 30 fps.

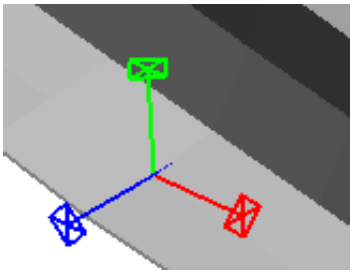
Symbol size: controlling ABP symbol display.

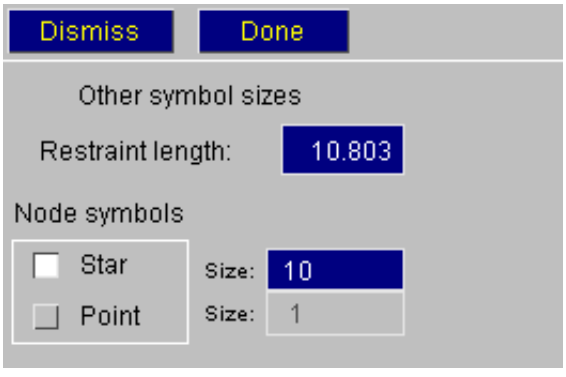
True radius x <*factor*> uses the radius reported for the current state multipled by <factor>. This is the default and is recommended for most cases. In all the analyses encountered so far particle radius has remained constant throughout the run, however it is written as a value at every state so this may change in the future.

Fixed radius <*value*> uses the stipulated fixed radius size.

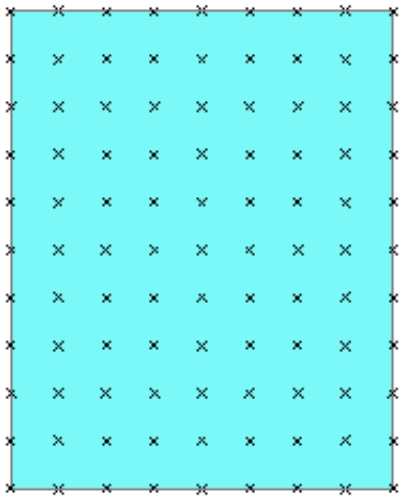
9.11 **Other Symbols**: controlling further symbols

Restraint length is the size given to restraint and constraint symbols applied to nodes in model space units.

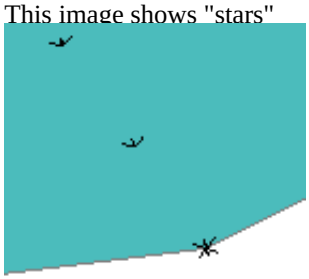




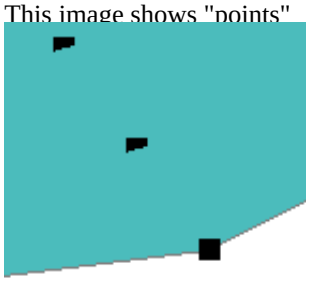
Node symbols controls how nodes are shown. Normally they use a star, but these can be quite slow to draw on some hardware, and points will render many (up to 100x) faster.



This image shows "stars"



This image shows "points"



This image shows a panel of shells with the default "star" symbol seen head on. The images on the right enlarge a corner detail to show both "star" and the "point" alternative.

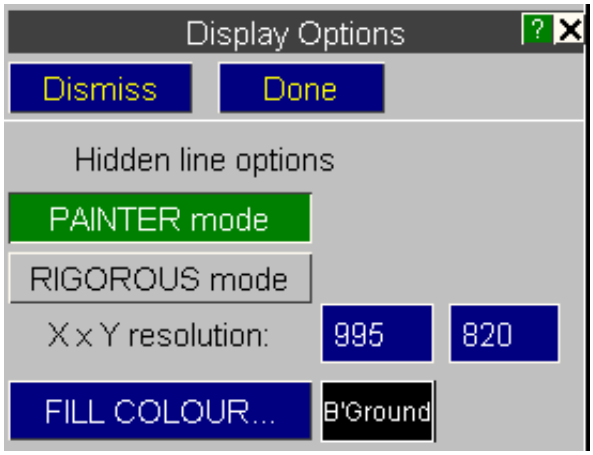
9.12 HIDDEN_OPTIONS... menu: Setting hidden-line display options.

This menu controls the algorithm and resolution used for creating hidden-line plots when in 2D mode. (It has no influence on 3D mode plots.)

It also controls the element fill colour used for hidden-line plots.

See:

PAINTER and **RIGOROUS**	To set 2D hidden-line calculation method
X x Y Resolution	To control the resolution used for calculating 2D hidden-line plots.
FILL_COLOUR...	To control hidden-line fill colour (all modes)



PAINTER and RIGOROUS: Setting 2-D hidden-line removal algorithm

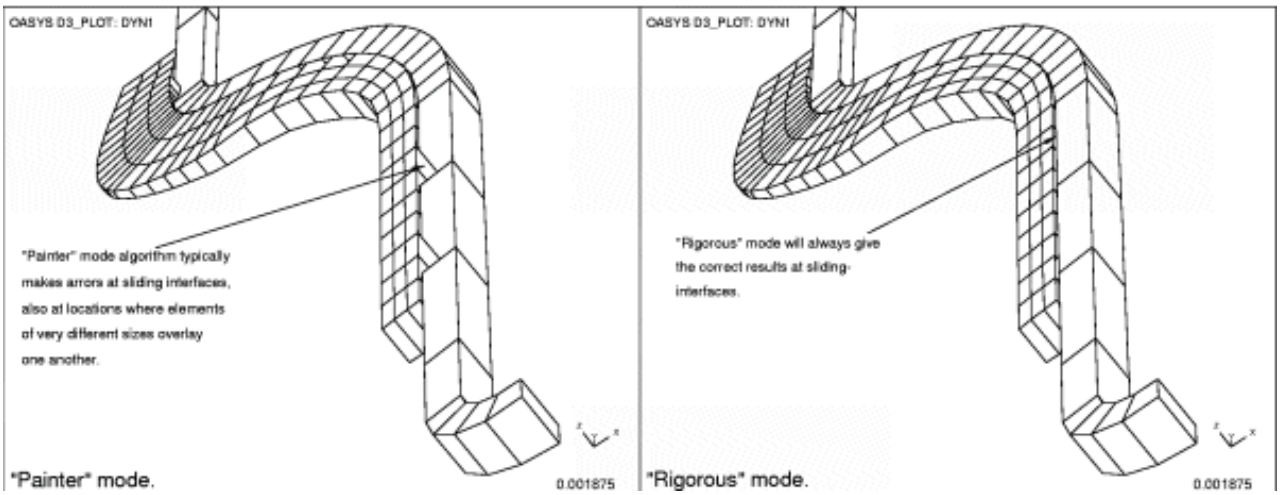
D3PLOT offers these two alternative hidden-line algorithms that are biased towards speed and accuracy respectively. These commands are irrelevant in 3-D mode.

PAINTER Where the image is broken into facets and these are drawn in order of increasing distance from the eye of the observer. This is the default method because it is quick and simple, but it can make mistakes: especially at interfaces, and it assumes that facets don't cross or intersect.

RIGOROUS Where the image is drawn internally, each pixel is checked to ensure that the correct parts are drawn, and only then is it mapped to the screen. This takes longer but will never make mistakes, hence it is offered as an alternative for problem plots.

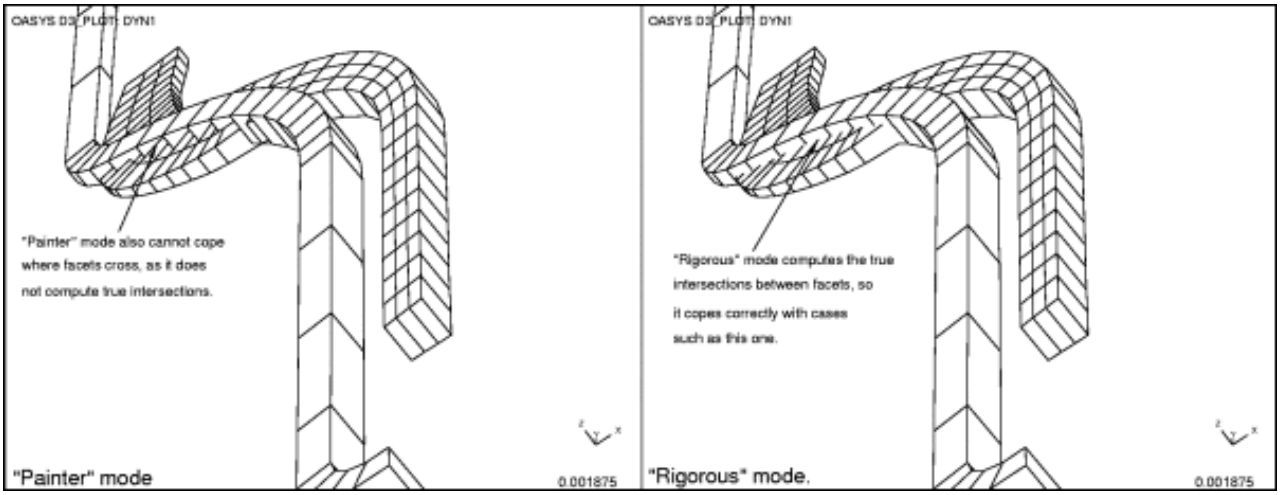
The two problems that can occur in "painter" mode, and the way that "rigorous" mode solves them, are shown in the following four figures.

"Painter" mode makes mistakes where large and small facets are adjacent


OASYS D3_PLOT: DYN1
"Painter" mode algorithm typically makes errors at sliding interfaces, also at locations where elements of very different sizes overlay one another.
"Painter" mode.
0.001875
OASYS D3_PLOT: DYN1
"Rigorous" mode will always give the correct results at sliding interfaces.
"Rigorous" mode.
0.001875

The figures above show how "painter" mode gets confused when rendering adjacent facets of different sizes: it draws them in the wrong order. "Rigorous" mode fixes this.

"Painter" mode cannot resolve the intersection of crossed facets


OASYS D3_PLOT: DYN1
"Painter" mode also cannot cope where facets cross, as it does not compute true intersections.
"Painter" mode
0.001875
OASYS D3_PLOT: DYN1
"Rigorous" mode computes the true intersections between facets, so it copes correctly with cases such as this one.
"Rigorous" mode.
0.001875

These two figures show how "painter" mode cannot compute the intersection when facets are crossed, whereas "rigorous" mode handles this correctly.

Which algorithm should I use?

Both algorithms are fully operative for all display modes that require hidden-line removal or its equivalent (eg **CT** Continuous-Tone, etc), but the "painter" mode will always be faster, which is why it is the default. You will only need "rigorous" mode when the errors in rendering shown in the figures above become unacceptable.

Users on 3D devices are not faced with this dilemma: the hardware Z-buffering used in hardware is, in effect, the "rigorous" algorithm. So you get the better method anyway.

X x Y resolution: Setting the 2D hidden-line calculation resolution

Both types of hidden-line calculation method use an internal resolution which, by default is set to the resolution (in pixels) of the graphics window. You can override this value by typing in new <x> and/or <y> values, and subsequent plots will use those values until you resize the window.

You might wish to reduce the resolution in order to speed up hidden-line plots, especially in "rigorous" mode, where the time taken is proportional to the product of <x> * <y>. There will be a corresponding reduction in image quality.

Hint: There is an easier way of doing this. Before you issue the plotting command resize the graphics window to make it small, draw the image, then resize it again to its original dimensions. The image will be computed at the smaller resolution but displayed at the full size.

Notes on hidden-line methods and resolution:

- Laser plots are computed at the current screen resolution so you could set a high resolution to improve laser image quality on a low resolution device.
- On windows devices the resolution is reset every time the window is resized: this operation will supersede any setting made explicitly with the **X x Y resolution** command.
- When dithered contouring or shading is used, or a solid filled plot (ie **CT**, **SH**, **SI**) drawn in "rigorous" mode, the display has to be calculated at the window pixel resolution, regardless of any of the settings made here. These internal settings are made for you automatically, and are transient for the duration of that plotting operation only, they do not change anything set here.
- All of the above is applicable to 2D mode plotting only. Z-buffered hidden-line plotting under OpenGL in 3D mode is implemented completely differently, it is implicitly "rigorous"

FILL COLOUR .... Setting the hidden-line fill colour for 2d and 3d elements.

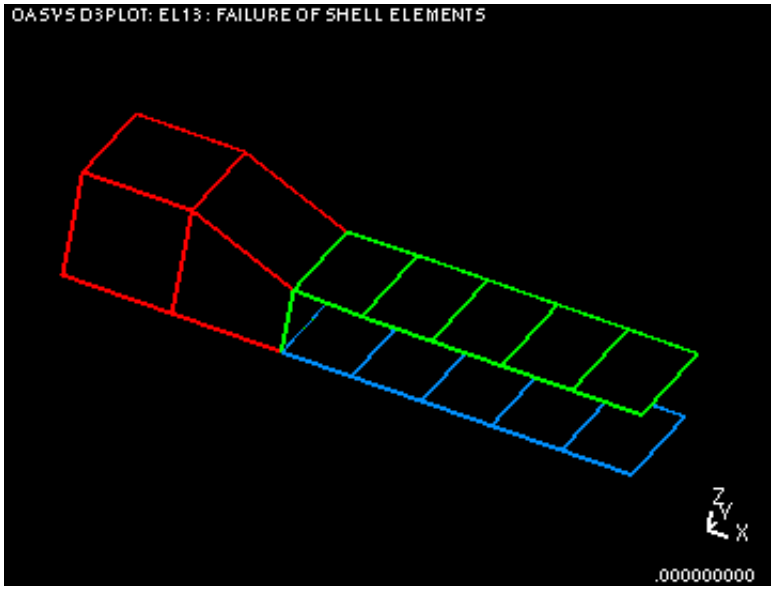
By default 2D and 3D elements are filled with the current background colour, whatever that may be.

However by clicking on the **FILL COLOUR...** button you may choose any other colour. The feedback button shows the current colour (here "B'Ground").

The following three images demonstrate how the fill colour can be modified:

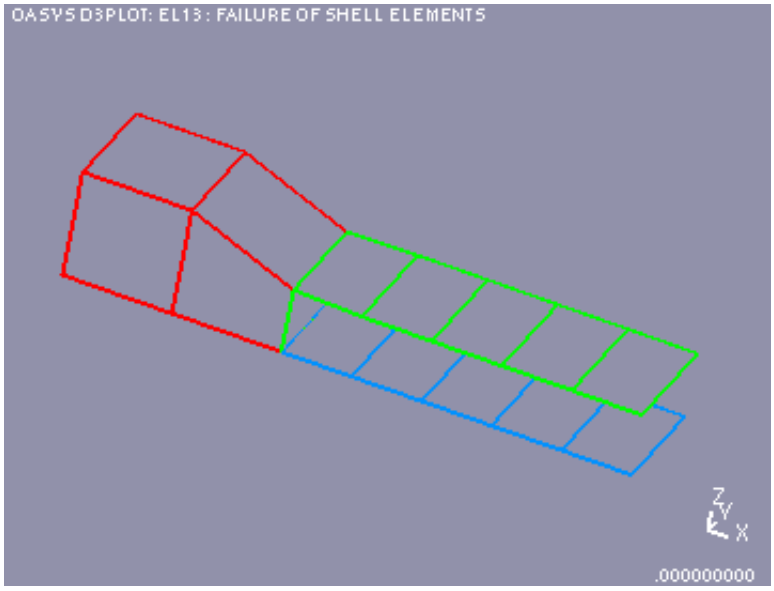
(1) This image shows the default behaviour:

Black background with element infill in the background colour.



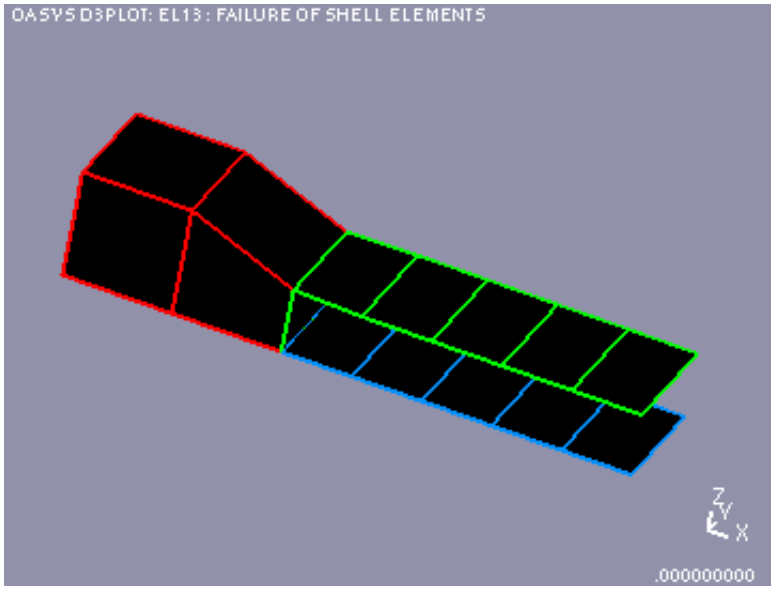
(2) In this image the background colour has been converted to grey.

The default infill is still background colour, so elements are also filled in with grey.

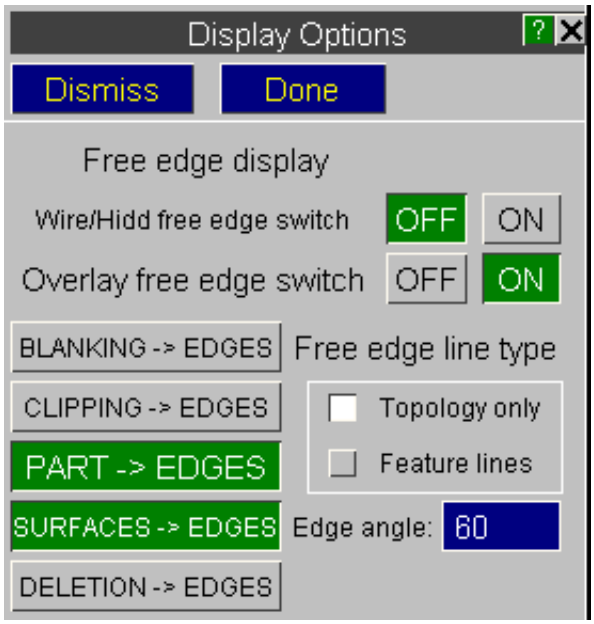


(3) In this image the background is still grey.

The hidden-line infill colour has been set explicitly to black.



9.13 FREE_EDGES... menu: Controlling free edge display of element borders



LI line and **HI** hidden-line plots, and the hidden-line overlay of element borders normally draw all element edges. In "free-edge" mode you can choose to draw only those borders which are "free" edges.

Two separate options are provided for controlling the display of free edged in Line/Hidden plotting and in all other plotting modes. The defaults for these 2 options are:

Line/Hidden	Defaults to ***off***
Other overlays	Defaults to ***on***

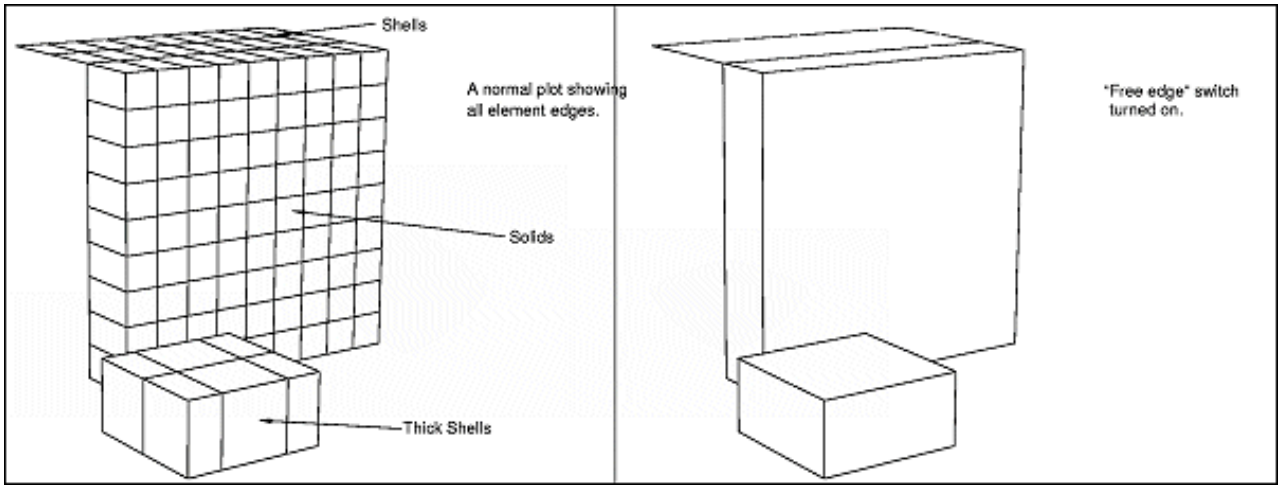
"Free" edges can be displayed either as topological free edges or as "feature" lines: this is explained in more detail below.

Overlays apply to all plotting modes that are not wireframe of hidden, and overlay display itself is controlled in the **OVERLAY** panel. (See Section 4.2.4)

Individual element overlay styles can also be set in the **PROPS** panel (Section 4.2.2.4).

What is a "free" edge?

A purely topological "free" edge is defined as an element (or face) border that is attached to only one element. This is illustrated in the two figures below.

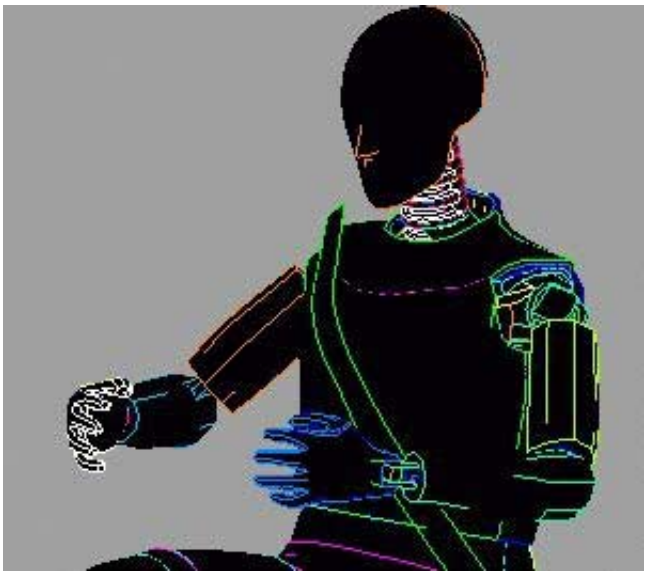


What is a "feature line"?

The purely topological definition of a free edge can sometimes give unsatisfactory images since it is dictated purely by element connectivity, and not by the actual shape of the mesh.

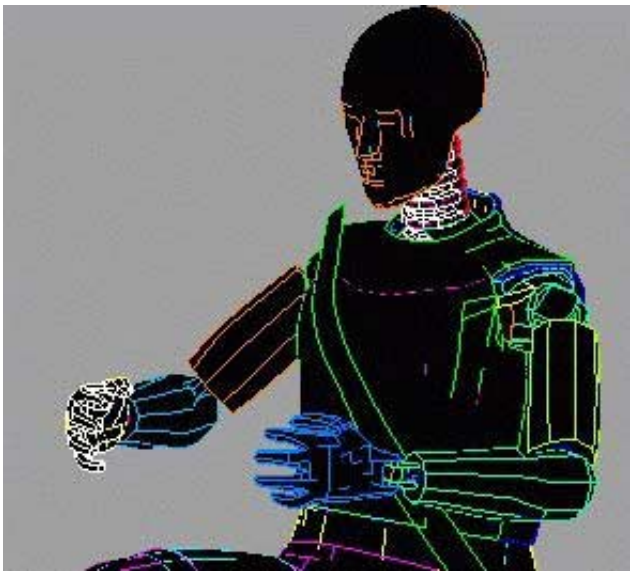
A "feature line" occurs when adjacent elements, or faces of adjacent 3D elements, have outward normal vectors that are more than the "edge angle" apart. This has the effect of inserting extra lines where the mesh changes shape, giving a better idea of the underlying surface.

The "edge angle" is the same as that used to denote sharp edges during smooth shading: see the Edge Angle notes in the section on lighting.



Free edge overlay

In this image purely topological free edges have been used. The image is understandable but a lot of detail is absent from areas of mesh of the same part id.



Feature line overlay

In this image feature lines with an "edge angle" of 20 degrees have been used. This has resulted in lines appearing in previously empty areas: in particular the mouth and eyes have acquired some detail, and the arms are better defined.

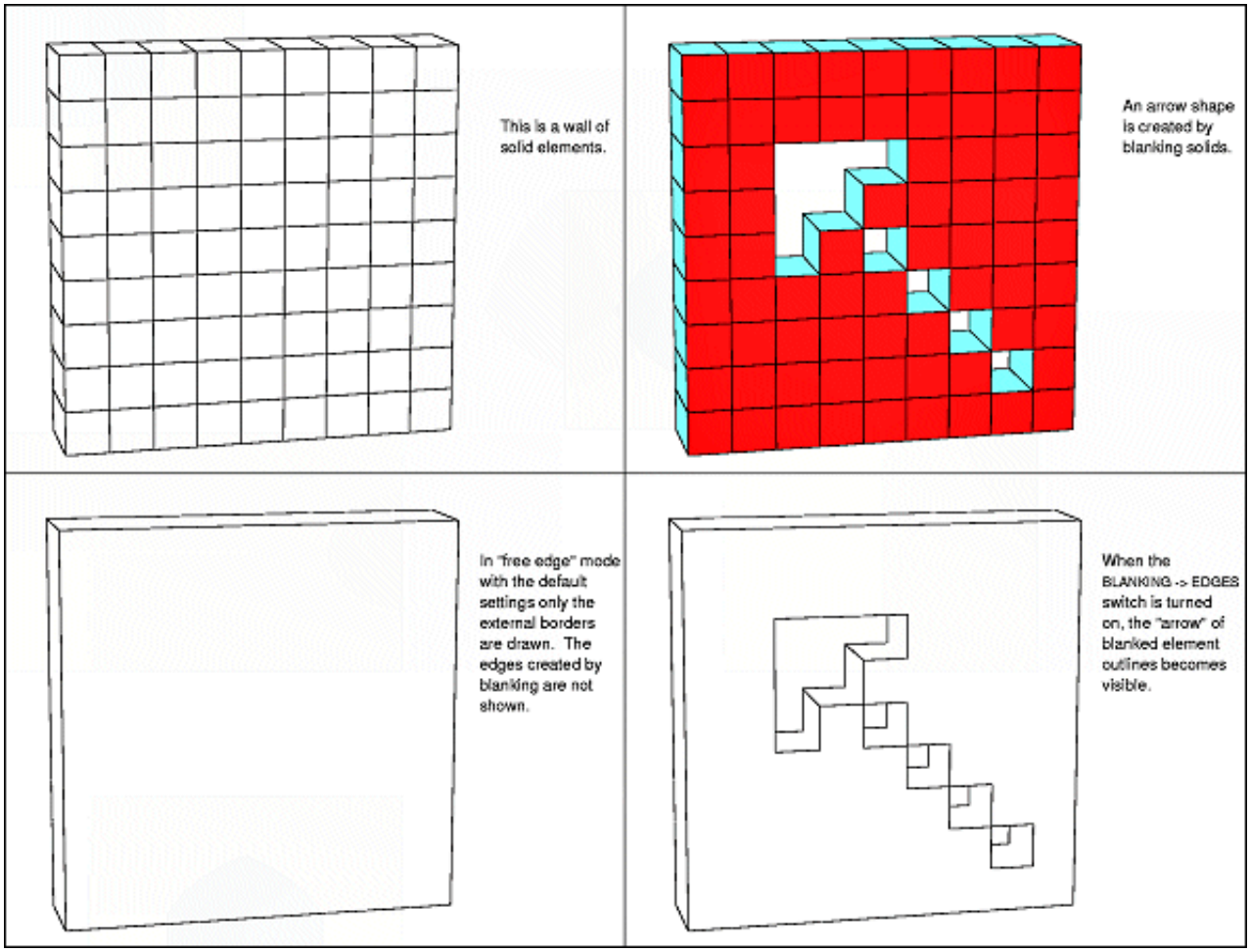
Feature lines are logically ORed with free edges when selected, and they can be used both as overlay on shaded/contoured plots, and as the edging mode for wire/hidden plots. They normally involve more vectors than pure free edges, so they take longer to draw and may have an adverse effect on animation speed.

Modifying the topological definition of a "free" edge

The definition above is open to modification: blanking elements can create free edges, as can volume-clipping and the borders between element materials (or contact surfaces). You can control whether or not each of these categories apply by setting the following switches:

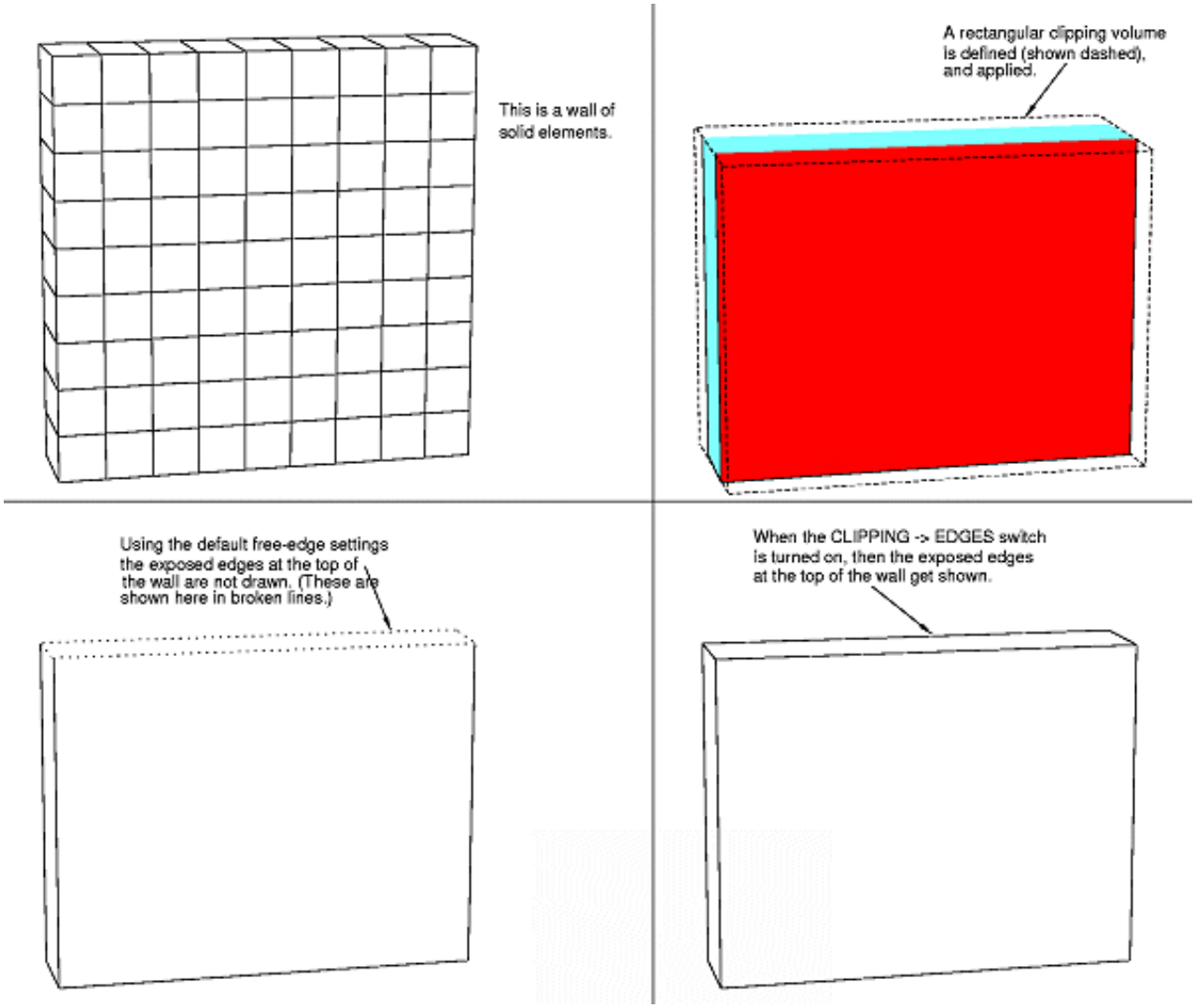
BLANKING -> EDGES Influence of Blanking on free edge display.

Normally blanking elements does not create "free" edges, but if you turn the **BLANKING** -> **EDGES** switch on then the edges created by blanking are shown. This effect is shown in the figure below


This is a wall of solid elements.
An arrow shape is created by blanking solids.
In "free edge" mode with the default settings only the external borders are drawn. The edges created by blanking are not shown.
When the BLANKING -> EDGES switch is turned on, the "arrow" of blanked element outlines becomes visible.

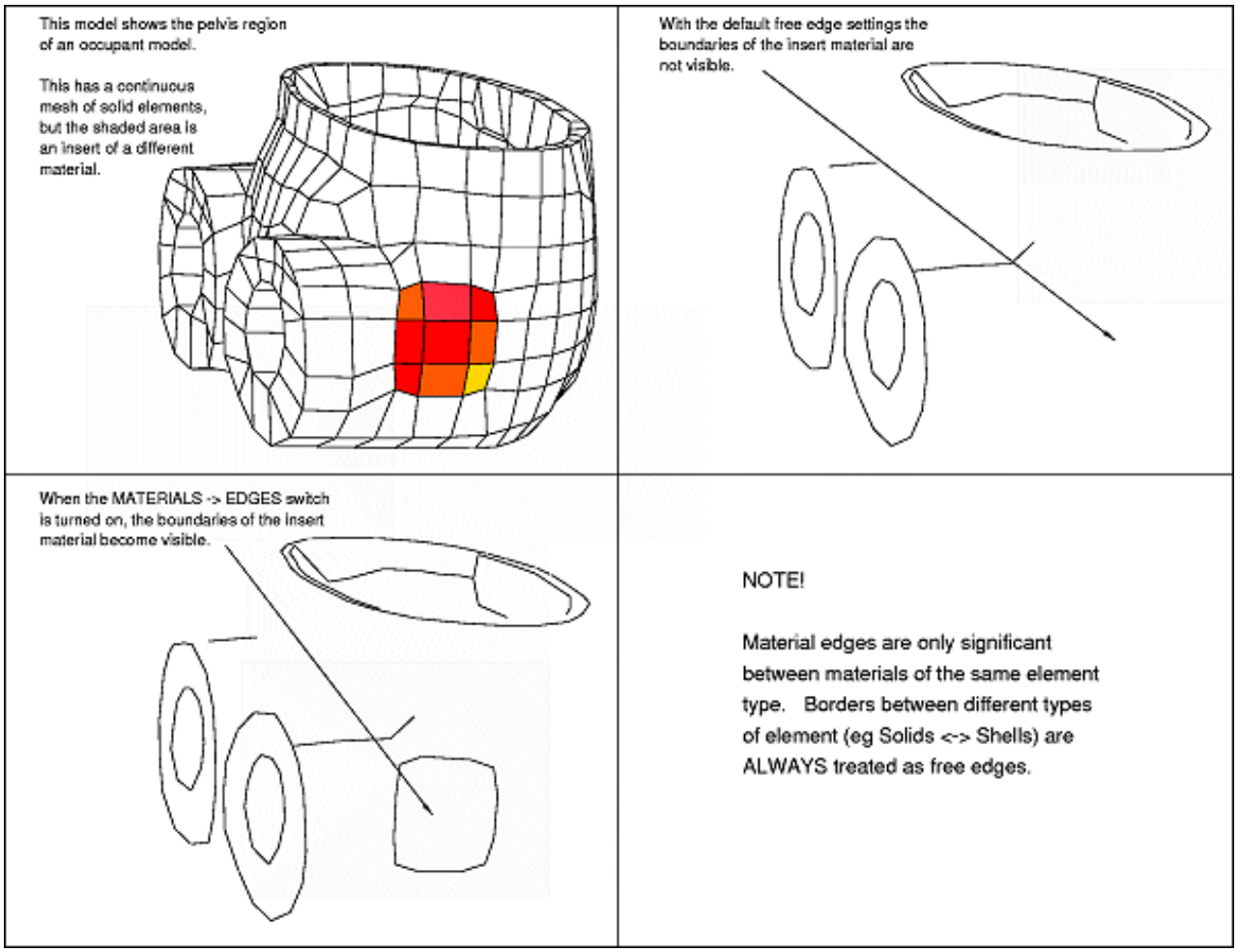
CLIPPING -> EDGES Influence of Volume-Clipping on free edge display

In the same way that element borders created by blanking do not, by default, create free edges; those exposed by volume clipping do not either. This is illustrated in the figure below where a clipping volume is used to remove solid elements from the same wall as in the previous example.


This is a wall of solid elements.
A rectangular clipping volume is defined (shown dashed), and applied.
Using the default free-edge settings the exposed edges at the top of the wall are not drawn. (These are shown here in broken lines.)
When the CLIPPING -> EDGES switch is turned on, then the exposed edges at the top of the wall get shown.

MATERIALS -> EDGES Influence of material boundaries on free edge display

The border between materials of the same element type does not, by default, qualify as a free edge. (Borders between different element types always qualify as edges.) If **MATERIALS -> EDGES** is turned on then such borders are treated as free edges. This is illustrated in the figure below.


This model shows the pelvis region of an occupant model.
This has a continuous mesh of solid elements, but the shaded area is an insert of a different material.
With the default free edge settings the boundaries of the insert material are not visible.
When the MATERIALS -> EDGES switch is turned on, the boundaries of the insert material become visible.
NOTE!
Material edges are only significant between materials of the same element type. Borders between different types of element (eg Solids <-> Shells) are ALWAYS treated as free edges.

SURFACES -> EDGES Influence of contact surface boundaries on free edge display

You can think of contact surface boundaries in the same way as material boundaries: the different surface ids equate to the different material ids, and the same edge definition logic applies.

However: The default setting of the **SURFACES -> EDGES** switch is on. The reason for this is that it would be very unusual to have two adjacent contact surfaces forming a single (geometric) surface, but it is quite common to have two or more coincident surfaces. In the coincident case it is useful to see the edges.

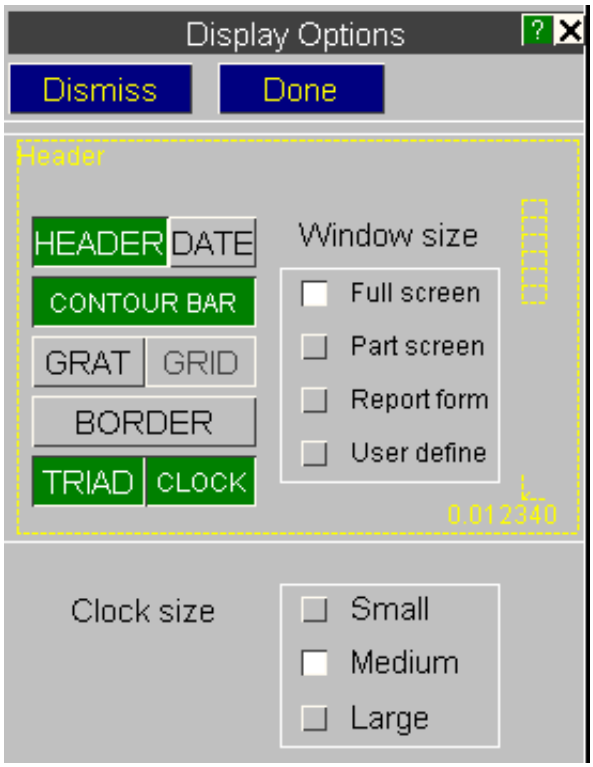
Notes on **FREE_FACE** options:

- Free edges are only computed for 2 and 3-D entities, that is: solids, thin shells, thick shells and contact surface facets.
- Free edge computation is worked out separately for each element type. For example an edge common to a shell and a solid will still be treated as a free edge.
- Where elements are deleted, due to material failure, free edges may be created at that and subsequent complete states.
- A "line" mode plot with free edges only is the fastest way of drawing something in D3PLOT since it combines minimal computation with only a small amount of screen vectors. It is a good way of assembling a quick animation
- Precomputed free-edge dynamic viewing is available: see Section 5.1.0.

9.14 **WINDOW_DRESSING...** menu: Controlling screen appearance.

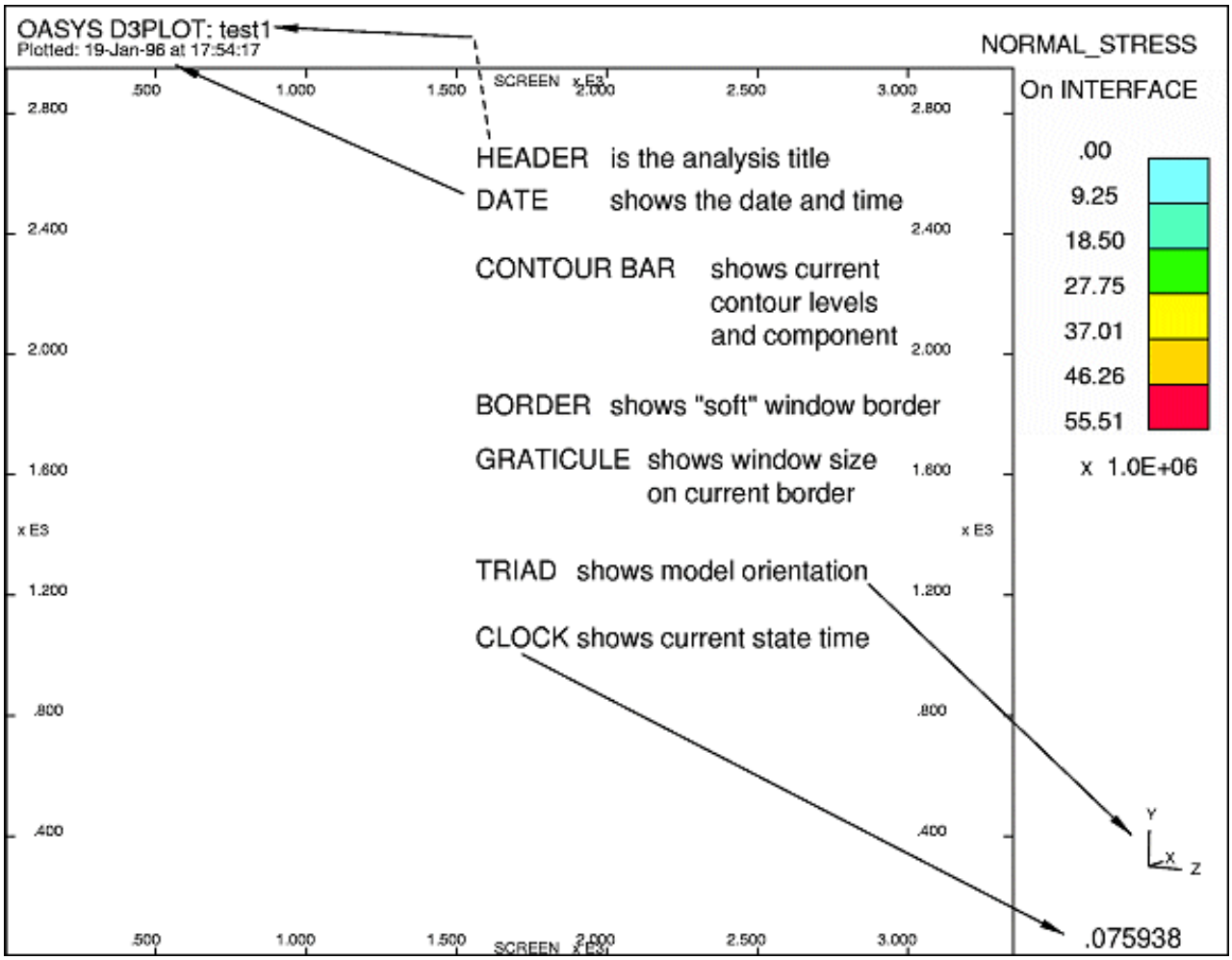
The "dressing" that can be added to plots: date, header, triad, etc is controlled from this window.

It also controls the "soft" window size.



Controlling "Window Dressing": HEADER, DATE, BORDER, CLOCK, TRIAD, CONTOUR_BAR

This figure shows the main "Window Dressing" attributes. Each of these can be switched on or off at will, the default being everything except the **GRATICULE** and **BORDER** are on. The **Window Size** shown in this example is **Part Screen** so that the borders are visible, but the default is **Full Screen**.



These options are all straightforward, with the exception of the **GRATICULE**. This gives the current model dimensions, but the space system used depends upon the current view:

Model space Is used if the view is orthogonal down one of the screen X, Y or Z axes. The appropriate XY, YZ or XZ coordinates are shown, and these move as the model moves (try dynamic translation and you'll see).

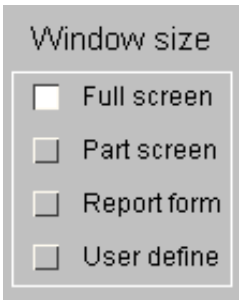
Screen space Is used if the view is not orthogonal. This just shows the current window dimensions (X = 0 - 4095, Y = 0 - 3129). It is only useful for setting up volume clipping using screen space orientation.

If a **GRID** is added it draws a grid on the screen at the current tick mark interval.

Controlling the "soft" window size: Window Size

By default the graphics can extend over the complete window area, and no "soft" window clipping is in force. You can restrict the graphics to a sub-set of the screen, ie to a "soft" window, by settings a **Window Size**.

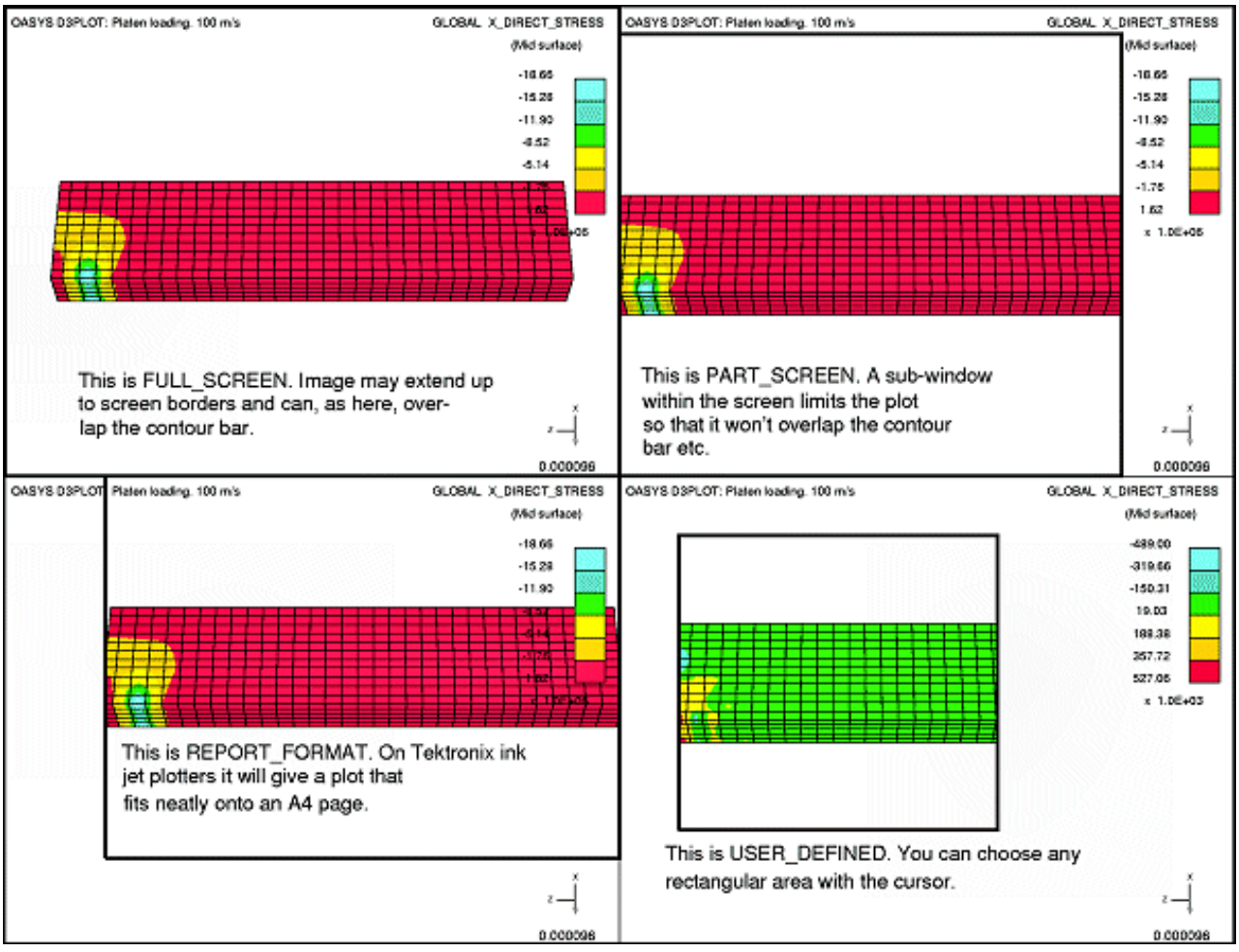
This is controlled by the radio buttons in the middle of the **Window Dressing** control panel, as shown here.



There are three pre-programmed options:

Full screen	(Default) use the whole screen
Part screen	Leave contour bar and header clear
Report format	Set dimensions for local hard-copier
User defined	Set screen rectangle using the mouse

The meaning of each of these is shown in the figure below:


This is FULL_SCREEN. Image may extend up to screen borders and can, as here, overlap the contour bar.
This is PART_SCREEN. A sub-window within the screen limits the plot so that it won't overlap the contour bar etc.
This is REPORT_FORMAT. On Tektronix ink jet plotters it will give a plot that fits neatly onto an A4 page.
This is USER_DEFINED. You can choose any rectangular area with the cursor.

9.15 Graphics text factor

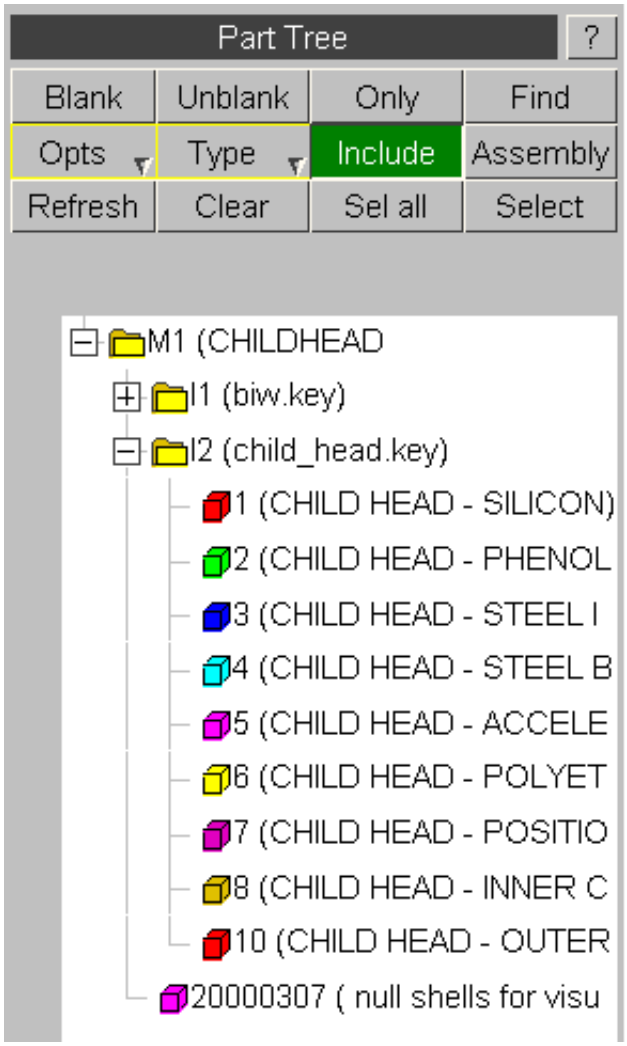
This is a simple factor in the range 1 to 10 applied to graphics text on plots, which will be scaled by the relevant amount.

Because not all hardware supports scalable fonts, and some larger fixed size fonts may not be installed, it is possible that some maximum text size on the display will be reached and a further increase in factor has no effect.

10 PART TREE

This enables quick navigation around a Model and helps with blanking etc. The Part Tree is available from the tab:

The part tree defaults to a view of the parts within the mode. If a ztf file is present, the model hierarchy by INCLUDE file will be displayed.

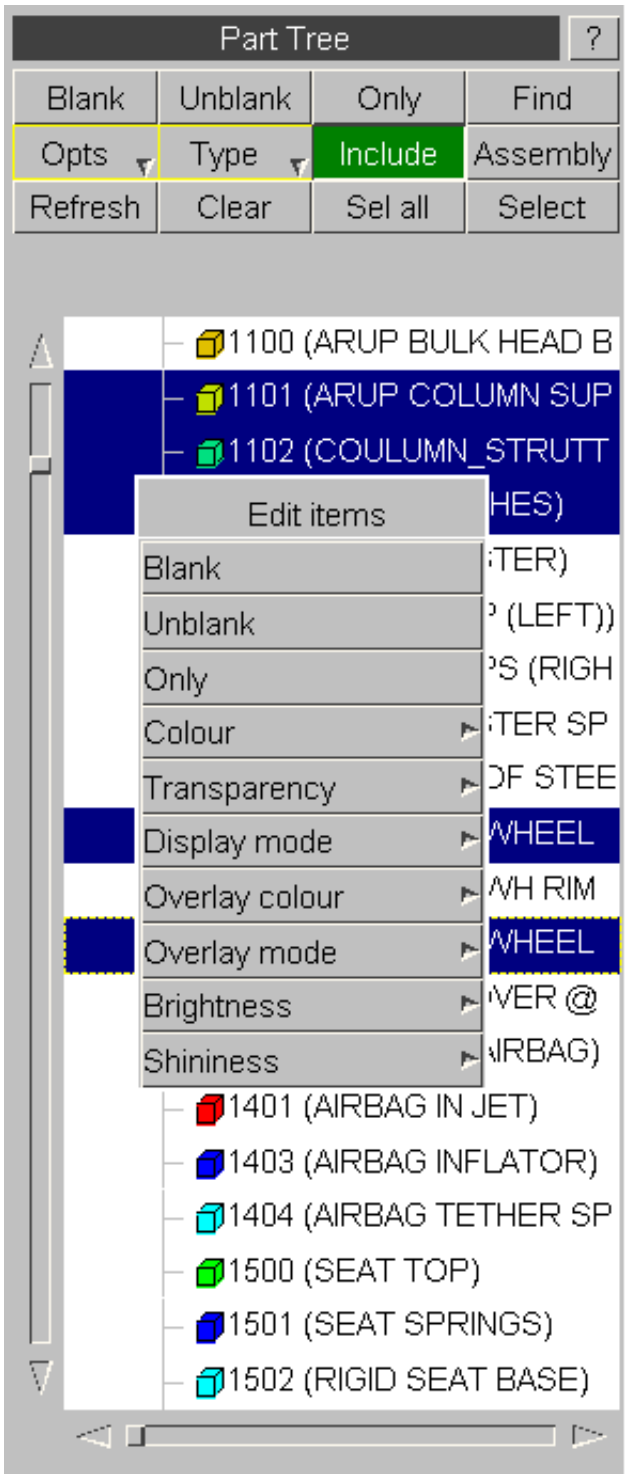


10.1 Part Tree Behaviour

Items can be selected by left-clicking anywhere on their row. Where selecting more than 1 item would be valid you can hold <ctrl> whilst clicking to select multiple items. Alternatively the <click> (start of range) .. <shift><click> (end of range) method (cf Windows) may be used.

Clicking on the [-] button next to models / include files / assemblies will collapse branches. Collapsed branches will have a [+] button which when clicked will expand the branch.

Right-clicking on an item or a selection of items produces a pop-up menu with the options shown on the right (not all of these options will be available for some selections).



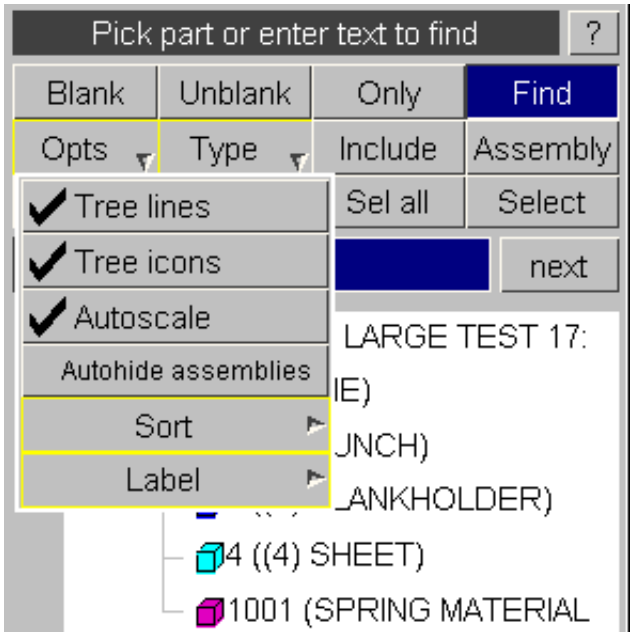
Blank	Blanks the item
Unblank	Unblanks the item
Only	Blanks all other items and unblanks the item
Colour	Colours the items (or elements associated with the item)as selected
Transparency	Sets the transparency the items (or elements associated with the item)as selected
Display Mode	Marks the item to be drawn with a particular method (wireframe, hidden, shaded or current)
Overlay Colour	Sets the item to be draen with a particular colour overlay
Overlay Mode	Sets the style of Overlay the item is to be drawn with (see section 4.2.2.2)

Brightness	Controls how light or dark the colour of the item is when illuminated, but the effect is to add matt colour (not whiteness, which would make it look shiny).
Shininess	Adds white highlights, but no colour, to make the object look shiny.

10.2 Part tree top menu bar

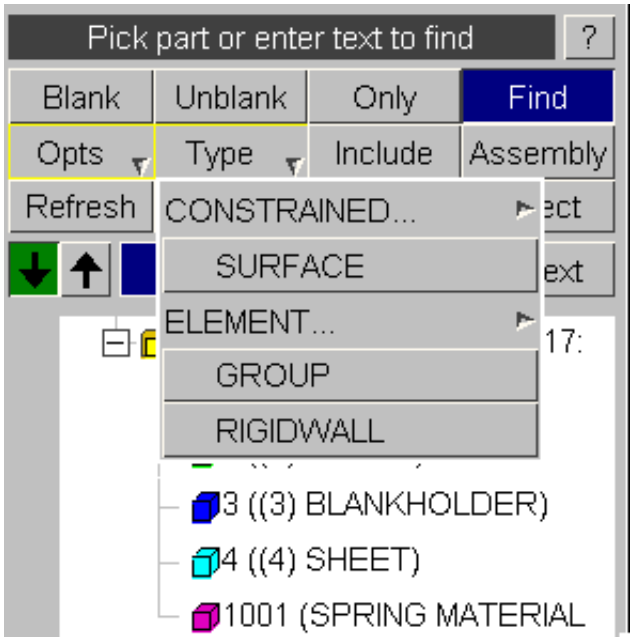
The top menu bar allows quick access top common functions, as well as controlling how the part tree behaves and is displayed.

Opts



There is a range of options for controlling the part tree available via the **Opts** pop-up menu. These include how items are labelled and ordered as well as whether assemblies, tree lines and icons are drawn.

Type



From the Type pop-up menu it possible to select a variety of different item types to be displayed in the tree in addition to parts. These appear below the parts in the tree, and most of the options (edit, blank etc) are available through the "right-click" menu.

Blank / Unblank / Only

One use of the part tree is as easy way access to blanking commands. **Blank**, **Unblank** and **Only** (blank all other items) and commands can be applied to the currently selected items.

Sel all / Clear

The **Sel all** and **Clear** buttons can be used to select all items and empty the selection respectively.

Select

The **Select** button invokes an object menu for selecting parts. Selection can also be made via the Quick Pick option "Locate in Tree". Operations such as blank,colour, etc may then be carried out on the selected entities.

Include/Assembly

The Include and Assembly buttons determine what type of hierarchy is displayed.

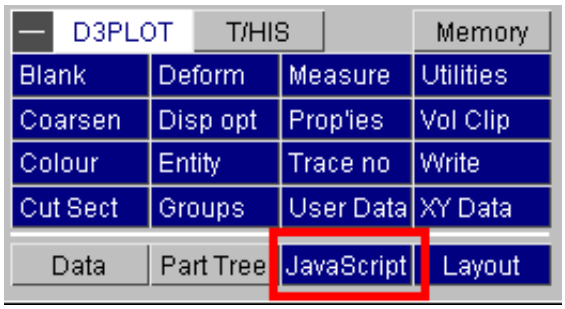
These have effect only when the requisite information is available to D3PLOT via the ztf file, and in the case of assemblies, via an assembly file written out by Primer.

Assemblies are user-defined hierarchical groupings of parts, created in Primer.

Find

The **Find** button gives a search option. Text or an ID number is entered in the text field. D3PLOT finds a part whose title contains the text, or a part with an ID matching the number. The arrows determine whether the search direction is up or down from the current selection. **Next** will find the next matching item. The search will only find matches for currently enabled options (i.e. if id is disabled and items are labelled by name only a search for part 15 will return no matches regardless of its presence in the tree).

11 The Javascript Interface



Programming D3PLOT externally for both batch and interactive use.

11.0 Introduction

Javascript is a freely available scripting language that is normally found performing the "work" behind interactive web pages, however its syntax and structure also make it an excellent tool for providing an externally programmable interface to programmes in general.

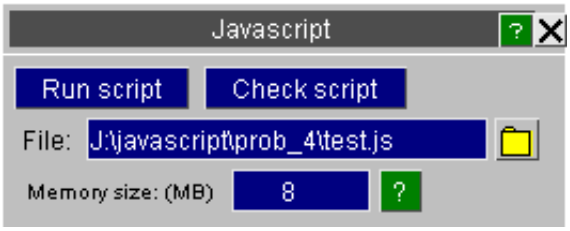
Within D3PLOT it is implemented as follows:

- There is a D3PLOT Application Programming Interface (API) which provides a range of functions that allow you to interrogate the database, open windows, generate plots, and so on. This is written in a very simple and non-intimidating way, with relatively few functions, that should be easy for non-programmers to use.
- There is also a function which issues "command line" instructions to D3PLOT, making it possible to use the code's full repertoire of command-line commands, meaning that virtually every function in D3PLOT is callable from within a Javascript.
- There is a special class of "user defined binary (UBIN) data components" that can be created from within a Javascript, making it possible to generate an unlimited number of new data components which then become available for processing in exactly the same way as the standard ones found in an LS-DYNA database.
- Finally the D3PLOT API shares the same user-interface methods and commands as the PRIMER one, making it possible to generate user-defined panels as part of the graphical user interface, and to read and write files.

Anyone familiar with C or shell script programming will find existing Javascripts are instantly readable, and can be given minor edits without further ado. For those who are more ambitious a good guide to the language is "**Javascript, A definitive Guide**" by David Flanagan, published by O'Reilly, ISBN 0596101996.

The sections below describe how to run Javascripts in D3PLOT, and summarise its Javascript API. For details of the API and its functions, and also some examples, see the Javascript Appendix

11.1 Using Javascript in D3PLOT.



Human-readable Javscripts need to be ***compiled***, meaning turned from something human-readable into a set of instructions that a computer can understand; and then ***run*** in their compiled form. They can be changed and rerun in their modified form at any time without having to exit and re-enter D3PLOT, making the "write, test, modify, re-test" development cycle very quick and easy.

11.1.1 Compiling and Running a script

Run Script	will both compile and run the script unless it contains syntax errors, in which case it stops with an error message when compilation fails.
Check Script	only compiles the script, reporting any errors found, and does not run it.
Memory size	is the threshold size of the Javascript memory "arena" at which Garbage Collection will take place.

11.1.2 Dealing with errors in scripts

Script errors come in two forms:

Syntax errors

Are mistakes of Javascript grammar or spelling, resulting in error messages during compilation.

These are easy to detect and correct since the line number and offending syntax are both described by the compiler. The script needs to be edited to correct the problem and then recompiled. Sometimes several iterations of the compile/edit cycle are required to eliminate all errors from a script.

Run-time errors

Are errors of context or logic in scripts that are syntactically correct, and thus have compiled, but which fail at some stage when being run.

A typical example of a run-time error is an attempt to divide a value by zero, yielding the illegal result infinity. More subtle errors involve passing an invalid value to a function, accessing an array subscript that is out of range, and so on.

The D3PLOT Javascript API has been written in such a way that it handles "harmless" run-time errors by issuing a warning and continuing execution, but that more serious errors which could result in the wrong answers being generated issue an error message and terminate.

Here is an example script which demonstrates both types of error. This script lists all the shell elements attached to the first node in the model, and calls to the Javascript API are hyperlinked to their relevant function definitions.

```
if(i = GetElemsAtNode(1, SHELL))
{
      Print("Number of shell elements on node " + GetLabel(NODE, 1) + " = " +
i.nn + "\n");

      for(j=0; j<i.nn; j++)
      {
            k = j + 1;
            Print("Shell #" + k + " = " + GetLabel(SHELL, i.list[j]) + "\n");
      }
}
else
{
      Print("No shells at node " + GetLabel(NODE, 1) + "\n");
}
```

This initial script is syntactically correct, and on an example model writes the following to the controlling terminal:

```
Number of shell elements on node 1 = 4
Shell #1 = 31318414
Shell #2 = 31318415
Shell #3 = 31319004
Shell #4 = 31319006
```

If a syntax error is deliberately introduced by omitting the second bracket at the end of line 1, leaving the "if" statement incomplete, ie:

```
if(i = GetElemsAtNode(1, SHELL)
```

Then this produces the compilation error:

```
Error when compiling J:\javascript\demo.js: at line 2:
SyntaxError: missing ) after condition
```

Which is computer-speak for "you left out the closing bracket on that '**if**' statement".

If a run-time error is deliberately introduced by omitting the second argument (**SHELL**) to GetElementsAtNode(), making the first line:

```
if(i = GetElemsAtNode(1))
```

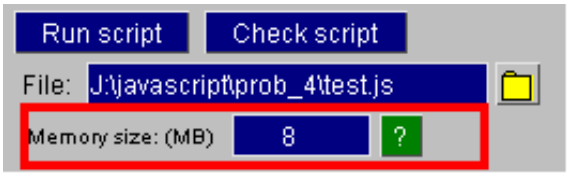
Then this is not picked up during compilation because the syntax is correct, but shows up when the script is run with the message:

```
%%% ERROR %%%
Fewer than 2 arguments supplied to Javascript function <get_elements_at_node>
```

And the script run terminates prematurely.

Because this example script has only one call to GetElementsAtNode() it is easy to identify and correct the problem, but in more complex scripts with many such calls it may be necessary to insert diagnostic Print() statements in order to track down a particular error.

11.1.3 Setting the Garbage Collection Threshold Size



(This is an advanced topic, and you don't need to understand it.)

Javascripts execute inside a memory "arena", allocated dynamically from the operating system, which grows in size as storage is requested within the script. This growth occurs due to requests for "new" variables within the script and also when API functions allocate and return values and objects, and it is limited only by what the operating system can deliver.

The nature of Javascript means that objects frequently become redundant, and it is wasteful not to reuse the storage that they occupy, therefore there is a "Garbage Collection" process running behind the scenes which periodically checks storage and releases that which is no longer needed. This process is automatic and hidden from the user, it just "happens".

However Garbage Collection is quite a CPU-hungry process, so it is only carried out periodically when the threshold set here is reached. This can sometimes be observed during script execution as a periodic "pause for thought", and if you are monitoring memory usage with a system tool you may see it drop during these pauses.

Clearly this threshold value must be large enough not to trigger excessively frequent (and costly) garbage collections, while at the same time not being so large that scripts build up large amounts of excess memory to the detriment of the rest of the programme. Experience has shown that the default of 8MBytes is a reasonable compromise between these two criteria for the majority of scripts, but you may find that scripts which allocate a lot of storage benefit from fewer "pauses", and hence run faster, if this value is increased.

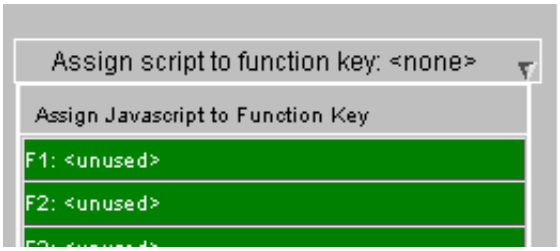
To recap:

- This threshold does ***not*** limit the memory the script can use, that is limited only by the operating system.
- It only sets the threshold at which Garbage Collection runs.
- Scripts which allocate a lot of memory, and which exhibit frequent pauses, ***may*** run faster with a larger value.

- ... and finally:

  ... if you don't understand this topic don't worry. Most scripts will run quite happily with the default value, and you can ignore this setting unless they appear to be struggling.

11.1.4 Assigning Javascripts to Function Keys



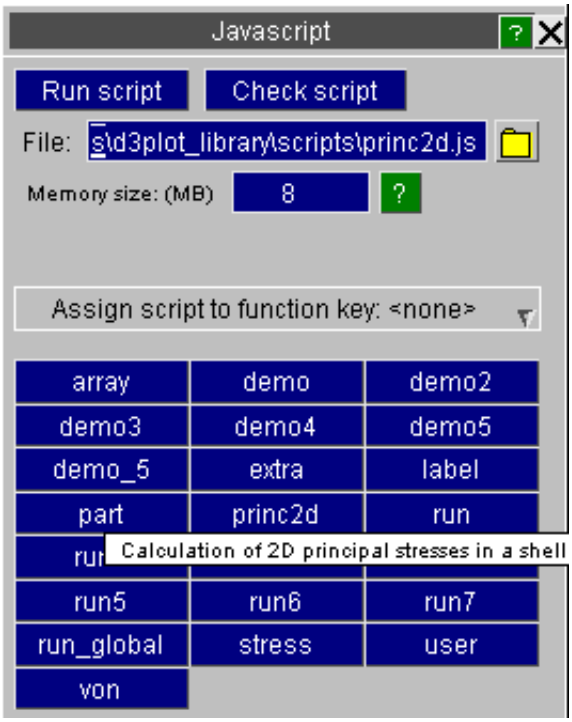
If a script is to be run repeatedly it can be convenient to set up a short-cut to it by assigning it to a function key, giving a "one click" method of running it.

Function keys can also be used to run D3PLOT command (.tcf) files, see Utilities, Function Keys, and key assignment may mix the two types at will. Javascripts so assigned should use the extension ".js" since this is how the two file types are distinguished from one another when the function key is used: files with any extension which is ***not*** ".js" are assumed to be command files.

Assignment of Javascripts to function keys can also be saved in the oa_pref file in exactly the same way as command

files see Utilities, Function Keys for details of how this is done.

11.1.5 Maintaining a library of Javascripts



It is also convenient to have a library of scripts in a defined location.

By default D3PLOT looks in **`$OASYS/d3plot_library/scripts`**, but you can define a different directory by setting the preference:

`d3plot*script_directory:`
`some_different_directory_name`

In the your oa_pref file.

All scripts found in the relevant directory will be listed in the Javascript panel, as shown in this example.

Using the "description:" comment at the top of a script to identify its purpose.

To help to identify scripts special comments are searched for in the top 10 lines of each script, and if **`description:`** is found, for example the comment line:

```
// description: Some description of the script's purpose
```

Then the description line is shown as hover text when the mouse is placed over that filename. In the example above the "**`princ2d`**" script has the line

```
// description: Calculation of 2D principal stresses in a shell
```

Using the "name:" comment at the top of a script to change its name

Normally the name shown for a script will be its filename, stripped of any leading pathname and trailing ".js" extension.

However if the string **`name:`** is found in the first ten lines of the script, then the following name will be used instead. For example the line:

```
// name: temporary
```

Will result in the script appearing with the name "**`temporary`**" in tha Javascript panel. This does not affect the actual name of the script, only the name on its library button.

11.1.6 Running a Javascript in "batch" mode.

All the above assumes that Javascripts will be run interactively from the user interface, however it is also possible to run a script in "batch" mode using the command line interface. The relevant command-line commands are:

`/JAVASCRIPT - +- COMPILE`		Compiles and checks the script, but does not run it.
	`+- EXECUTE`	(Re)compiles and runs the script
	`+- MEMORY <nnn>`	Resets the Garbage Collection threshold to <nnn> MBytes

To run a Javascript from batch these commands need to be placed in a command file and run using the command line "**-cf=***command filename*" option. For example the command file might be:

```
... some other commands
/JAVA EXEC my_script.js
...some further commands
```

And the command line required to run D3PLOT might be something like:

```
$OASYS/d3plot93.exe -d=default -cf=command_file -exit analysis_name
```

Obviously multiple script invocations may be placed in a command file. For more information see:

Command and Session files	Describes command files, and explains how to create and use them
Valid D3PLOT command line arguments	Describes the various command line arguments, and how to use them

11.2 The D3PLOT Javascript API

11.2.1 Summary table of API functions

The table below lists all the functions in the API, grouped by category. The hyperlinks will take you to their detailed descriptions in the Javascript Appendix.

...Window...	**Window and frame management**
CreateWindow()	Create a new window containing 1 or more models in <model list>
DeleteWindow()	Delete (a) window(s), optionally "disposing" of orphan models
SetWindowActive()	Sets the "active" flag on the specified window(s)
GetWindowMaxFrame()	Returns the highest frame in <window_id>
SetWindowFrame()	Displays frame <frame_number> in the specified window(s)
GetWindowFrame()	Returns the current frame of <window_id>
GetWindowModels()	Returns an object containing information about the model(s) in <window_id>
Get...	**"Getting" general information about the model**
GetNumberOf()	General routine to return the quantity of many different things
GetTime()	Return the analysis time of the current state, or of <state> if defined.
GetLabel()	Return the external label of internal <type/item>
GetPid()	Return the internal part id of internal <type/item>
GetMid()	Return the external material id of internal <type/item>
GetTopology()	Return an object containing topology, #nodes and part id for internal <type/item>
GetElemsAtNode()	Return a list of elements of <type> at <node>
GetData()	Return scalar, vector or tensor data for data <component> of <type/item>
QueryDataPresent()	Returns JS_TRUE if data <component> is present.
Set...	**"Setting" general information about the model**
SetCurrentModel()	Sets <model_id> to be current for data "get" and "put" operations
SetCurrentState()	Sets <state_id> to be current for data "get" and "put" operations
...Ubin...	**Manipulating User Defined Binary (UBIN) data components**
CreateUbinComponent()	Create a user-defined internal binary (UBIN) data component and return its "handle"
LocateUbinComponent()	Returns an object with the <handle> and other attributes of UBIN component <name>
DeleteUbinComponent()	Deletes UBIN component <handle>
GetUbinData()	Get data from UBIN component <handle> for <type/item>
PutUbinData()	Insert data for UBIN component <handle> for <type/item>

...Cut...	**Manipulating cut sections**
SetCutSection()	Defines cut-section attributes in <window_id>
GetCutSection()	Retrieves cut-section attributes from <window_id>
GetCutForces()	Returns the forces, moments, centroid and area from the cut-section in <window_id>
Dialogue...	**Command line dialogue insertion**
DialogueInput()	Executes 1 or more lines of command-line dialogue commands
DialogueInputNoEcho()	As for DialogueInput(), but with no echo to dialogue box
Miscellaneous	**Diagnostic and other general functions.**
Print()	Prints <arg 1> and any following arguments on the terminal (stdout).
Common	**User interface and i/o functions common with the Oasys PRIMER Javascript API**
File	Handles file opening, closing and general i/o
Window	Handles creation and management of menu system windows
Widget	Handles primitives and processing in Window classes

11.2.2 More information about using the Javascript API

The main documentation of the API is in the Javascript Appendix under the headings below. It is strongly recommended that you read this Appendix before embarking on writing Javascripts, as it explains the concepts and pitfalls in more detail.

Notes on programming

- Independence of scripts, and runs of scripts
- Argument types (integer, double, etc) used in the interface
- Compulsory and optional arguments in the functions above
- Return values from functions
- Execution errors and warnings

Notes on data abstraction and processing.

- Adapting programming style to improve memory efficiency
- Using User-defined Binary (UBIN) data components
- The "current" model and state during data manipulation
- Ordering processing for efficiencient data extraction
- Using Direct Disk Access (DDA) for "hopping around" a large dataset
- Internal item indices versus external item labels
- Ordering of data in Vector and Tensor arrays
- Testing for the presence or absence of a given data component
- Special considerations when processing adaptively remeshed analyses

Notes on handling windows and models in windows.

- The "current frame" in windows
- Multiple models in windows

Recommended window setup when using this API

Detailed Description of Javascript Interface Functions.

- Window manipulation
- Setting "current" status items
- Functions to "Get" and "Put" data and other information
- Functions for processing User-defined Binary (UBIN) data components
- Functions for processing cut-sections
- Functions for inserting command-line dialogue input
- Functions for diagnostic ouput

List of valid constants to be used in the functions above:

11.2.3 Examples

By far the easiest way to learn Javascript is by example and, more specifically by modified existing scripts to do what you want.

Oasys software comes supplied with examples in the **$OASYS/*programme*_library/examples** directory (for D3PLOT **$OASYS/d3plot_library/examples**) and you are free to use and modify these files for your own purposes.

There are also some simple documented examples in the Javascript Appendix Examples section.

12 MORE ABOUT DATA AND DATA COMPONENTS

12.0 Introduction to this section on data and data components.

It is important that you understand what data LS-DYNA writes, and how D3PLOT processes it for presentation. Therefore this section is organised to provide the following information:

12.0.1 Format and contents of the LS-DYNA databases.

In Section 12.1 the format and in Section 12.2 the contents of each of the LS-DYNA databases processed by D3PLOT are described in more detail. The various output switches available in LS-DYNA to control their contents are also described.

12.0.2 “Global” and “summary” data descriptions.

D3PLOT makes a distinction between global and summary data components, (those for the whole model, materials and contact surfaces), and components for individual element types. Sections 12.3 to 12.5 describe the "summary" data components available for the whole model, materials and contact surfaces respectively.

12.0.3 Data components for nodes and element types that write results.

Not all entity types write results to the files processed by D3PLOT. Only nodes, solids, shells, thick shells, beams and contact surface segments have data components available for them. These are described in Sections 12.6 to 12.11 respectively.

12.0.4 Data components for entities that do not write results.

Those entities which do not write results to databases processed by D3PLOT are lumped-masses, springs, seat-belt types, joints and stonewalls. Nevertheless geometric and other data components are available for these and are described in Section 12.12.

12.0.5 Theory behind data manipulations.

The theory and formulae behind the computation of derived data components is given in Section 12.13. For example von Mises stress, principal stresses, etc.

12.1 Format of the LS-DYNA databases processed by D3PLOT

D3PLOT processes three of the binary database file types from LS-DYNA:

Complete state **Dynamic relaxation** **Eigenvalue (ex Nike)**	**`.ptf`** } **`.rlf`** } ----	Contains basic geometry and topology of, and transient results for, nodes, solids, shells, beams and thick shells. Eigenvalue files contain modeshape (not transient) results.
Contact force	**`.ctf`**	Contains geometry and topology of, and transient results for, contact surface segments.
Extra time history	**`.xtf`**	Contains geometry and topology of lumped-masses, springs, seat-belt types, stonewalls and joints. (But the results for these are not processed by D3PLOT.) Note that the .xtf file is not supported by MPP dyna, and that it is increasingly likely to be supplanted by the "binout" (or LSDA) database file. From V9.0 onwards the **`.ztf`** file contains all the plottable data previously extracted from the .xtf file, so there is no loss of functionality in D3PLOT.
Pseudo time history	**`.ztf`**	Contains extra information culled from the input deck: nodal contacts, restraints and constrained, part and contact names. From V9.0 onwards also contains all static data previously saved in the **`.xtf`** file, making plotting of these extra items possible if an .xtf file is not present. The **`.ztf`** file is generated by Oasys PRIMER directly from the input deck, usually by running a batch translation phase immediately after the ls-dyna analysis.

12.1.1 The “familied” nature of database files.

All of the database files above use a family structure. There is always a "root" member, which may have between 1 and 999 "children". This is done to keep file sizes down: there are many advantages to having a few moderately big files instead of one huge one.

The "root" members:	Have the names **`<job>.ptf`**, **`<job>.ctf`**, etc.
The "child" members:	Have names **`<job>.ptf01`** to **`<job>.ptf99`**; then **`<job>.ptf100`** to **`<job>.ptf999`**

The maximum size of a family member is set when LS-DYNA is run, and a new "child" member is opened if writing the current block of information would cause the current member to spill over this limit.

By default 7Mbytes is used (in single precision), corresponding to 1835008 words of data, but it is possible to change this when the LS-DYNA job is submitted via the X= parameter on the DYNA submission line, or via the Oasys shell. However very large analyses often use a larger family size in order to stop results states spilling over into multiple family members.

By default D3PLOT determines the file family size automatically (by taking the greater size of the first two members of a family, and rounding up to the nearest Mbyte). You can over-ride this by setting an explicit size (see Section 4.1.7) but this should rarely, if ever, be necessary.

Note: Previous versions of D3PLOT used signed 32 bit integers to represent disk addresses, which limited total database size to 2^{31} words, or 2GWords.

Version 8.0 onwards of D3PLOT uses 64 bit integers to represent disk addresses, even on 32 bit machines, which means that it can access disk addresses up to 2^{63} words, or 9e18 words.

LS-DYNA from approximately 2002 (~LS960) onwards no longer adheres to the maximum database size logic for .ptf files for larger models. If the state is too big to fit into a single family member it is allowed to increase in size in order to contain the state. D3PLOT handles this automatically.

12.1.2 Setting the family member size of database files

The following strategy is adopted when the software is supplied to you by Oasys:

- The whole software suite defaults to 7MByte family members if no external environment variables are set.
- The environment variable **FAM_SIZE** may be set to a family size in MBytes. This should be an integer between 1 and 100, and it is the preferred method of changing this value since it will be picked up by all the software in the suite.
- In the **oasys** shell you can set a different size when submitting a LS-DYNA job with the **Binary file size** slider in the **ADDITIONAL FILES** panel.
- In D3PLOT the value defaults to **FAM_SIZE** if this has been set, otherwise it is computed automatically. You can also modify the value both at file input and during a session using the commands described in Sections 4.1.1 and 4.1.8.

Please contact Oasys if you need more advice on changing this value.

12.1.3 Handling missing family members

It is not possible to use any of the database files above if their "root" family member is missing. This contains control, geometry and topology data that is required if the rest of the file family is to be read. However it is possible to process families in which some children have been deleted.

The data written to a file family is: <Control and topology> <state #1> <state #2> ...

and certain rules are used to make selective removal of child family members easier.

(1) If adding a <state> would overflow the maximum size permitted in the current member, it is closed and a new one is opened.

(2) A new family member is **always** started following the writing of a restart dump file.

(3) If a <state> is too big to fit into a single member then the maximum size rule is still obeyed: the first member is filled to capacity, then the next one is opened to take the remainder of the data. Thus <states> will be written in pairs (or, if large enough, triplets, quadruplets and so on) of family members.

This is best explained by example. Two are given here: one for a moderate size analysis, and one for a huge one. In both cases a maximum family size of 7MB is assumed.

Example 1: Basic control and topology requires 2MB, each state uses 3MB.

Root member		Child #3	Restart dump => new member
Control + Geometry	2MB	<State #5>	3MB
<State #1>	3MB	<State #6>	3MB
Child #1		Child #4	
<State #2>	3MB	<State #7>	3MB (Last state in file)
<State #3>	3MB		
Child #2			
<State #4>	3MB		

Example 2: Basic control and topology require 8MB, each state 12MB.

Root member		Child #2		Child #4	
Control + Geometry	7MB	<State #1: part 1>	7MB	<State #2: part 1>	7MB
(part 1)					
		Child #3		Child #5	
Child #1		<State #1: part 2>	5MB	<State #2: part 2>	5MB
Control + Geometry	1MB			And so on in pairs	
(part 2)					

Note that the basic geometry spills into the first child, and that subsequent <states> always come in pairs of files. In this example child family members containing states could be removed, but only in matched pairs.

Hint: Use the UNIX command **ls -lt** to look at your files. This will give a "long" listing showing file size, and also sort them into chronological order of creation.

Then look for matched pairs of files that will, in this example, have 7MB and 5MB sizes with the smaller file being marginally more recent.

D3PLOT will skip gaps in a file family sequence if the **FILE_SKIP** environment variable is set. This is an integer that defines how many missing files will be skipped before the search is abandoned. The default value set in the **oasys_81** shell is 5, but values much larger than this (up to 999) could be used. Larger values will increase the time delay when **SCAN**ning files as children are searched for on disk. See Section 4.1.1 and 4.1.7 for ways to alter this value at run time.

12.1.4 Disk format of binary database files.

All the files above are "random access binary files". This means that they are stored in the binary format of the machine, and cannot be read or edited with tools such as a text editor.

There are several machine formats available, the most common one being:

IEEE (Stands for the Institute of Electrical and Electronic Engineers). This is the most common, and a de-facto standard on Unix workstations.

CRAY Used on Cray XMP, YMP, C90, J90, etc.

Convex This is now no longer current, but older C1xx, C2xx and C3xx machines used this format.

There are two word-lengths in common use:

32 bit Single precision, using 4 bytes. This gives 7 or 8 decimal figures of precision, and a decimal exponent in the range +/-38.

64 bit Double precision, using 8 bytes. This gives 14 or 15 decimal figures of precision, and a decimal exponent of +/-308 in IEEE format, or +/-2465 in Cray format.

There are two possible way to arrange the bytes in the words:

Big endian Byte order [4] [3] [2] [1] (single precision example)

(or **M**ost **S**ignificant **B**yte : **MSB**)

Little endian Byte order [1] [2] [3] [4] (single precision example)

(or **L**east **S**ignificant **B**yte : **LSB**)

Oasys LS-DYNA is normally supplied such that it writes 32 bit IEEE (Big endian) files, regardless of the machine architecture and precision it is running on. This is achieved by converting results if necessary in the output routines.

D3PLOT is capable of reading any of the formats above. It detects the format by scanning the contents and converts it automatically on input to the native format of the machine on which it is running. Therefore you will sometimes see a message like:

```
[This machine is Cray 64 bit Normal endian ]

[File format is IEEE 32 bit Normal endian ]
```

when it reads files. This is just a notification message and you need not take any further action: in the example above they will be converted automatically to 64 bit Cray format.

Does using 32 bit format on a 64 bit machine reduce the precision of my answers?

It does not affect the precision of the ***calculation*** at all: this is always performed at the full precision of the machine, and dump files also use the full precision (so restarts do not compromise accuracy). It is ***only*** the databases written for post-processing that are truncated to 32 bits.

So yes, it does affect the precision in post-processing: you will be forsaking 14 or 15 significant figures for 7 or 8. But consider for a moment what this means: a typical car model might be 5 metres long, so single precision output will resolve displacements to somewhere between 0.05 and 0.5 microns. And anyway, is your calculation really accurate to 0.00001%?

It is hard to envisage the situation where this matters and, in the opinion of Oasys, the benefits of halving disk space usage and having files which may be post-processed without conversion on workstations far outweigh any disadvantages. However Oasys will supply the code with 64 bit output on request.

Are there any other limitations in using 32 bit file formats?

Only one: if you are post-processing on a 64 bit machine you will have problems if your 32 bit file contains integers (eg node, element or material numbers) outside the range $+/-2^{24}$ (16,777,216). This is because the bit patterns of integer and floating point numbers become hard to tell apart at this point, and this distinction is required for automatic conversion.

So, if you run on (say) a Cray ***and*** post-process on that machine then try to keep your node, element, material and other labels below this limit. If you take your files to a 32 bit machine for post-processing the problem will not arise and you can use the full valid integer range.

Alternatively, please request a version that writes 64 bit output from Oasys.

Why does D3PLOT sometimes mix up IEEE and Cray 64 bit formats in older files?

Prior to LS-DYNA 940 it was not possible to tell if a 64 bit file came from a Cray or an IEEE machine. (Subsequent database files contain values which make distinction possible.)

Therefore if the file type is indeterminate D3PLOT defaults to one or the other format, which may be wrong. You can control the default 32 and 64 bit file types using the following environment variables:

Variable name	Word size	Possible values	Default in V8.0
FILE_TYPE_32	32 bit	**IEEE, CONVEX**	**IEEE**
FILE_TYPE_64	64 bit	**CRAY, IEEE, CONVEX**	**IEEE** (was **CRAY** in V7.x)

eg **setenv FILE_TYPE_64 CRAY** Would be required to read an older 64 bit Cray file in V8.0

Next Section

12.2 Contents of the LS-DYNA database files processed by D3PLOT

(See Section 12.2.4 for a summary table of all contents in all files.)

12.2.1 The "complete state" (**.ptf**) file. (Also **.rlf** and **d3eigv** files.)

12.2.1.1 Defining output parameters in LS-DYNA

The following LS-DYNA control cards control output of this file:

***DATABASE_BINARY_D3PLOT**	Is mandatory. Controls output frequency.
***DATABASE_EXTENT_BINARY**	Is optional. This card allows switching of certain parts of the file's contents on/off. These are discussed in section 9.2.1.3 below.

12.2.1.2 The generic contents of the complete state file

Control information:		
	Number of nodes	
	Number of each type of element	
	Number of materials	
	Amount of data written for each element type	
Basic geometry & topology:		
	Undeformed nodal coordinates	
	Element topology	
	Arbitrary numbering tables	
Complete state:		(Repeated for all times dumped)
	Time	
	Global data: energies, velocities, masses, stonewall forces	
	Nodal coordinates	
	Temperatures at nodes	
	Nodal velocities	
	Nodal accelerations	
	Solid element stresses and plastic strain	
	Thick shell stresses and plastic strains	
	Beam forces	
	Thin shell stresses, strains & related data	
	Deleted element tables	

12.2.1.3 Controllable contents of the complete state file.

The options on the ***DATABASE_EXTENT_BINARY** card in the LS-DYNA input deck allow you to control the following contents of the complete state file.

NEIPH "Extra" data components for solid elements. Default **None**.

Some material models generate more information for solids than can be written in the standard formats. For these "extra" variables may be written. They are <neiph> scalar values that follow the normal data. They:

- Are written for every solid element, regardless of material model.
- Have no explicitly defined component names associated with them: this will depend on the material model.

NEIPS "Extra" data components for shells and thick shells. Default **None**.

As with solids some material models write extra information for shells. They are **<neips>** scalar values that:

- Are written at every "surface" output for every shell, regardless of material model used.
- Have no explicitly defined components associated with them: this will depend on the material model.

MAXINT Number of "surfaces" written for shell and thick shell elements. Default **3**.

The default value of 3 writes data at neutral axis, innermost and outermost integration points for shells and thick shells. Values other than 3 write results for the first **<maxint>** integration points: ***read Section 12.8.2.2 before using them.***

STRFLG Write directional strain tensors for solids, shells & thick shells. Default **Off**.

By default no strain tensors are written for any elements, (although effective plastic strain is). Turning this flag on causes the strain tensors for solids, shells and thick shells to be written. Note that:

- A single tensor at the element centre is written for solids.
- Tensors at innermost and outermost integration points ***only*** are written for shells and thick shells, regardless of the **<maxint>** value.

The following flags can be used to reduce database size by controlling the output for shells and thick shells. By default they are all **On**.

NOTE: ***None of the following four flags influences the output for solid elements.***

SIGFLG Controls the output of the stress tensors for shells and thick shells.

The symmetric stress tensor (6 values) is written at **<maxint>** "surfaces" for every shell and thick shell, so turning this off usually saves 18 values per shell. You will, of course, then not be able to post-process any stresses.

EPSFLG Controls the output of effective plastic strain for shells and thick shells.

The effective plastic strain (1 value) is written at <maxint> surfaces for every shell and thick shell. Turning this off will usually save 3 values per shell.

RLTFLG Controls the output of force and moment resultants for shells.

Force and moment resultants (8 values) are written for every shell (but not for thick shells). Turning this off will save 8 values per shell.

ENGFLG Controls the output of thickness and strain energy density for shells.

Thickness, strain energy density and two other (unused) values are written for shells (but not for thick shells). Turning this off will save 4 values per shell.

CMPFLG Composite material stress output in local axes. Default **Off**.

By default all stress tensors are written in the global coordinate system. Turning this flag on causes those from composite materials to be written in the material local axis system(s).

WARNING: ***There is no way for D3PLOT to tell from the database that these stress tensors are in the local system. It assumes that ALL STRESS TENSORS ARE IN THE GLOBAL SYSTEM.***

If you use this facility IT IS YOUR RESPONSIBILITY to interpret your results correctly: they will be reported in D3PLOT as global stresses.

BEAMIP Number of "extra" data values written for beam elements. Default **None**.

All beam elements write 6 basic forces and moments, but certain element and material formulations can generate extra data:

Resultant formulation (Belytschko-Schwer) beams can generate a further 15 "plastic" values, depending upon the material model used.

Integrated (Hughes-Liu) beams can generate 5 "extra" stress and strain values at each integration point.

There is no way of telling from the database whether "extra" beam results contain "Integrated" stress/strain data, or "Resultant" plastic data.

LS-970 and earlier: inconsistent beam sign conventions.

There is an inconsistency of sign convention in beam output from versions of LS-DYNA prior to release 971.

- "Resultant" (typically Belytschko-Schwer) elements use one sign convention
- "Integrated" (typically Hughes-Liu) elements use the opposite sign convention for 4 of the 6 output components.

The following table shows the status quo ***up to and including LS-DYNA release 970***:

Component	Matching?
Fx	Same
Fy	Opposite
Fz	Opposite
Mxx	Opposite
Myy	Opposite
Mzz	Same

Which is right?

Sadly there is no "right" for beam output, as different users have different conventions. The confusion arises because of the different ways in which the beam types work: integrated beams have integration points at their centre, whereas resultant beams have (potential) hinges at their ends. The former reports force in the beam, and the latter reactions at the supports.

This has serious implications when plotting beam data (see section 4.2.5) and when extracting cut section forces and moments through beam structures (see section 6.4.6)

LS-971 onwards: beam sign conventions corrected.

At some stage during the development of LS-DYNA 971this bug was corrected, and the output from resultant and integrated beams now match. The sign convention that has been adopted is the "integrated" one.

D3PLOT handling of Beam sign convention problems.

Unfortunately it is not possible to determine the analysis code version accurately from a database file (at the time of writing, September 2008, output databases from LS-DYNA 971 still report their version number as 970). Therefore D3PLOT adopts the following approach:

- Contour plots of beam data are always shown "as is". This tends not to matter since it is usually easy enough to see what plots mean.
- However when computing cut-section forces through beams the sign convention is vital, therefore D3PLOT will prompt you for the sign convention to be used.

IT IS YOUR RESPONSIBILITY to interpret your beam results correctly.

DCOMP Data compression flag. Default **1.**

The default value of "1" means that database compression is turned off and a full set of data values is written to the database for each rigid element in the model. If this option is set to "2" data values will not be output for rigid elements. This option can significantly reduce the size of the binary files written by LS-DYNA if the model contains a large proportion of rigid elements.

The following output options are currently not supported by D3PLOT, and should not be changed from their default (off) states:

SHGE Output of shell hourglass energy.

STSSZ Output of shell element time steps.

12.2.2 The Extra Time History (**.XTF**) file

12.2.2.1 Defining output parameters in LS-DYNA

The following LS-DYNA control cards control output of this file:

***DATABASE_BINARY_XTFILE** Is mandatory. It controls output frequency and if you don't have this card you won't have an **.XTF** file.

The contents of this file are determined by LS-DYNA, and you have no control over them.

It is intended primarily for time-history processing, but D3PLOT uses it to obtain the geometry and topology of springs, seat-belt types, joints, stonewalls and lumped masses. D3PLOT ignores the state information in it (use Oasys T/HIS to process that).

12.2.2.2 Generic contents of the **.XTF** file.

Control data:	
	Title
	Number of nodes
	Number of elements
	Amount of data written for elements
Initial data block:	
	Nodes for reactions
	Spring topologies
	Seat-belt etc topologies
	Lumped mass data
	Joint geometry
	Stonewall geometry
Arbitrary numbering tables	
Beam element time history blocks	
Complete states (repeated n times)	
	(Not processed by D3PLOT)

Next section

12.2.3 The Contact Force (**.CTF**) file

12.2.3.1 Defining output parameters in LS-DYNA

The following LS-DYNA control cards control output of this file:

***DATABASE_BINARY_INTFOR** Allows you to set an output frequency for this file different to that used for the **.PTF** file, which is the default. (Note that you must request this file when submitting the job in order for it to be written.)

It is strongly recommended that you DO NOT use an output frequency for this file that is different to that used for the .PTF file. If you do D3PLOT can have problems trying to synchronise contact surface data with other results.

This file contains the topology and results for all segmented contact surfaces in the model. You have no control over its internal structure.

12.2.3.2 Generic contents of the **.CTF** file

Control information:	
	Number of nodes
	Number of interface surface and segments
Geometry and topology:	
	Undeformed nodal coordinates
	Interface segment topology
Arbitrary numbering tables	
Complete state data: (repeated n times)	
	Time
	Global data: Energies and velocities
	Current nodal coordinates
	Nodal velocities
	Interface segment stresses
	Nodal contact forces

12.2.4 Key to finding information in OASYS LS-DYNA database files

	.PTF file	**.CTF file**	**.THF file**	**.XTF file**
Whole model data:				
Global energies	KE, IE, TE	KE, IE, TE	KE, IE, TE	KE, IE, TE
Global velocities	VX, VY, VZ	VX, VY, VZ	VX, VY, VZ	VX, VY, VZ
External work				EW
Time-step			DT	

Data by material:

Material energies	KE, IE	KE, IE	
Material velocities	VX, VY, VZ	VX, VY, VZ	
Masses	Mass	Mass	

Nodal data:

Geometry	[X,Y,Z] coords	[X,Y,Z] coords	[X,Y,Z] coords	[X,Y,Z] coords
Displacements	[dX,dY,dZ]	[dX,dY,dZ]	[dX,dY,dZ]	[dX,dY,dZ]
Temperatures	[temps]		[temps]	
Velocities	[vX,vY,vZ]	[vX,vY,vZ]	[vX,vY,vZ]	
Accelerations	[aX,aY,aZ]		[aX,aY,aZ]	
Nodes for reactions				Geom + reacts

Solid & thick shell element data:

Topology	Connectivity & mat'l	Connectivity & mat'l
Stresses &c	Stress tensor	Stress tensor
	Plastic Strain	Plastic strain
	Strain tensor	Strain tensor
	Extra variables	Extra variables
		(Els in TH blocks)

Thin shell element data:

Topology	Connectivity & mat'l	Connectivity & mat'l
Stresses &c	Stress tensor	Stress tensor
	Plastic Strain	Plastic strain
	Strain tensor	Strain tensor
	Extra variables	Extra variables
	Force/moment resultants	Force/moment resultants
	Thickness & strain energy density	Thickness & strain energy density
		(Els in TH blocks)

Beam element data:

Topology	Connectivity & mat'l	Conn'ty & mat'l	Conn'ty & mat'l
Results	Forces & moments	Forces & moments	Extra data Mat'l 29
	Extra data Mat'l 29	(Els in TH blocks)	(Els in TH blocks)

Spring element data:

Topology	Conn'ty & mat'l
Results	Force/elongation
	Moment/rotation

Seat-belt etc elem data:

Topology	Conn'ty & mat'l
Results	Force/elongation
	/pullout etc

Lumped-mass element data:

Topology	Node no & mass

Stonewall data:

Geometry			All geom
Normal forces	Fwalls	Fwalls	Fwalls

Interface data:

Geometry	All geom	
Summary forces		Fx,Fy,Fz summaries
Detailed results	Segment stresses	
	All nodal forces	

Joint data:

Topology	Nodes, type stiffness

Airbag (control volume) data:

Results	Pressures etc

Notes on file types:

.PTF file :	"Complete state" file	is written at low frequency and has a complete description of the whole model.
.CTF file :	"Contact force" file	is written at the same frequency as the **.PTF** file, & is a complete description of the interface forces & stresses.
.THF file :	"Time history" file	is written at a high frequency and contains a (user-specified) subset of the model for detailed examination. It is used for XY graph type display (eg T/HIS).
.XTF file :	"Extra time-hist" file	is written at the same frequency as the **.THF** file, and contains extra information that cannot be accomodated in the **.THF** file structure. Used for XY plots, and also for providing the geometry for the graphics display of walls, joints, springs, etc
.ZTF file :	"Extra static data" file	is not generated by LS-DYNA, but rather by running Oasys Primer either in batch (immediately after the analysis) or manually. Extra "static" data are extracted from the keyword input deck and become available for post-processing.

12.3 Global (whole model) data components

This section describes the data components available for the "whole" model. These can only be processed as numerical values in **WRITE** and **XY_DATA**.

Global data components are all extracted from the complete state (**.PTF**) file.

Raw components written by LS-DYNA are shown **THUS**, those calculated or derived by D3PLOT are shown oblique **THUS**.

Energies

KE_KINETIC_ENERGY	The total kinetic energy.
IE_INTERNAL_ENERGY	The total strain and other non-kinetic energy.
TE_TOTAL_ENERGY	The sum of the two terms above.

Velocities

VX_X_VELOCITY	Average X velocity of the whole model
VY_Y_VELOCITY	Average Y velocity of the whole model
VZ_Z_VELOCITY	Average Z velocity of the whole model
VR_VELOCITY_RESULTANT	Vector sum of above (computed by D3PLOT)

Mass values

MASS	The sum of all material masses.

Note: This only contains mass from solid, shell, beam and thick shell materials. Lumped masses, stonewalls, and any other entity types which might contribute mass to the model are not included. Thus, for most models, this is an underestimate.

Momentum values (From **`mass * velocity`**)

MX_X_MOMENTUM	Mass * average X velocity
MY_Y_MOMENTUM	Mass * average Y velocity
MZ_Z_MOMENTUM	Mass * average Z velocity
MR_MOMENTUM_RESULTANT	Vector sum of the above

Note: The mass used here is the computed mass above, so the computed momenta may be an underestimate.

Next section

12.4 Part ("material") data components

This section describes the data components available for materials. These can be processed as numerical values in **WRITE** and **XY_DATA**, and from release 9.3 onwards part-based data may also be plotted as well. (In this case all elements of the part are "contoured" in the same colour representing their part value.)

Material data components are all extracted from the complete state (**.PTF**) file.

12.4.1 The "material" data available depends upon the files present.

Complete state files (.ptf)	These files are always present, and contain part data for: 1. All parts (materials) of solid, shell, thick shell, beam and SPH elements. 2. All nodal rigid bodies 3. All discrete and seatbelt elements. Category (1) may be extracted and processed numerically (in **WRITE** and **XY_PLOT**), and visually (in 2D3D plotting) because the part numbers of these element classes are known. Category (2), nodal rigid bodies, may be listed as a category under **WRITE, GLOBAL**; however they cannot be plotting because their underlying nodes are not known. Category (3), discrete and seatbelt element part data, cannot be processed from .ptf files alone because these files do not contain information about these elements.
ZTF files	If a .ztf file has been written from Oasys PRIMER then it becomes possible both to visualise discrete and seatbelt elements, and also to determine their part numbers. Therefore it is possible to associate part data with elements for these types, and process them in the same way as solids, shells, etc. This also means that discrete and seatbelt elements can be selected and processed "by part" for operations such as blanking, picking, and so on.
XTF files	If an .xtf file is present, which will increasingly not be the case, as it is not supported by MPP versions of LS-DYNA, then discrete and seatbelt elements can be visualised; but the file does not contain the information required to associate these element types with their part ids as defined in the LS-DYNA keyword input deck, so their part-based data cannot be accessed.

D3PLOT automatically "culls" from part listings, menus, etc those parts which either do not contain any elements or for which the element type cannot be determined. The only time that they become visible is when a **WRITE**, **MATERIAL_SUMMARY** listing is produced.

12.4.2 The data components available for Materials

Raw components written by LS-DYNA are shown **THUS**, those calculated or derived by D3PLOT are shown oblique **THUS**.

(Note that these are the same component names as are used for global (whole model) results.)

Energies

KE_KINETIC_ENERGY	The material kinetic energy.
IE_INTERNAL_ENERGY	The material strain energy.
TE_TOTAL_ENERGY	The sum of the two terms above.

Velocities

VX_X_VELOCITY	Average X velocity of the material
VY_Y_VELOCITY	Average Y velocity of the material
VZ_Z_VELOCITY	Average Z velocity of the material
VR_VELOCITY_RESULTANT	Vector sum of above (computed by D3PLOT)

Mass values

MASS	The reported material mass.

Momentum values (From **`mass * velocity`**)

MX_X_MOMENTUM	Mass * average X velocity
MY_Y_MOMENTUM	Mass * average Y velocity
MZ_Z_MOMENTUM	Mass * average Z velocity
MR_MOMENTUM_RESULTANT	Vector sum of the above

12.5 Contact Surface summary components

This section describes the summary (whole surface) data components available for contact surfaces, as opposed to those available for contact segments (which are described in Section 9.11).

These cannot be plotted, they can only be processed as scalar values in **WRITE** and **XY_DATA**.

Contact surface summary data components are all extracted from the contact force (**`.CTF`**) file. They are all forces, computed in D3PLOT by summing the nodal forces on each surface.

12.5.1 The data components available.

Forces

FX_CONTACT_X_FORCE	Total X force on the surface.
FY_CONTACT_Y_FORCE	Total Y force on the surface.
FZ_CONTACT_Z_FORCE	Total Z force on the surface.
FR_CONTACT_FORCE_RES	Vector sum of above.

12.5.2 Why results from the **`.CTF`** file may differ from those in the **`.XTF`** file.

The **`.CTF`** file is an instantaneous "snapshot" of the status of all contact surface segments at the time-step when the dump took place.

The **.XTF** file output is generated within LS-DYNA by averaging contact forces on each surface over the time period since the previous time-history dump. Therefore the results are not instantaneous.

There are pros and cons to both approaches: the averaging used for **.XTF** output means that very fast spikes are not missed, but their magnitude is attenuated by the averaging process; whereas the instantanous data in the **.CTF** file reports the correct magnitude but, in doing so, may miss a spike altogether. The **.XTF** approach is better for time-history plotting, and the **.CTF** approach for graphical output.

12.5.3 When the Master and Slave side forces differ.

Where the forces on master and slave sides are with 5% of one another the average value is reported. Where the difference is greater the side with the higher force magnitude is used.

Differences can occur - normally one side is zero, for example in the surface of single-surface contacts, or discrete nodes impacting a surface. (In the latter case the nodes are not on segments, so are not written to the **.CTF** file.)

12.6 Nodal data components

LS-DYNA writes nodal results to the **.PTF** and **.CTF** files. All directional results are in the global cartesian system except accelerations at nodes classified as accelerometers.

D3PLOT is able to plot nodal data on 2D and 3D elements by averaging across their faces.

It will process scalar nodal data, element data averaged at nodes and other geometrical data in the **WRITE** and **XY_PLOT** menus.

12.6.1 The nodal data components available from the **.PTF** file.

Undeformed coordinates

BX_BASIC_X_COORD **BY_BASIC_Y_COORD** **BZ_BASIC_Z_COORD**

Current coordinates

CX_CURRENT_X_COORD **CY_CURRENT_Y_COORD** **CZ_CURRENT_Z_COORD**

Displacements (Derived from **<current> - <undeformed>**)

DX_X_DISPLACEMENT ***DY_Y_DISPLACEMENT*** ***DZ_Z_DISPLACEMENT***

DR_DISP_RESULTANT

Velocities

VX_X_VELOCITY **VY_Y_VELOCITY** **VZ_Z_VELOCITY**

VR_VEL_RESULTANT

Accelerations

AX_X_ACCELERATION **AY_Y_ACCELERATION** **AZ_Z_ACCELERATION**

AR_ACCEL_RESULTANT

Temperatures

Note: Temperatures are only written in a thermal-only, combined structural and thermal, or structural analyses using thermal materials.

TEMPERATURE Coordinates, velocities and accelerations are not written in a thermal-only analysis.

12.6.2 The nodal components available from the `.CTF` file.

Contact Forces

XG_GLOBAL_X_FORCE **YG_GLOBAL_Y_FORCE** **ZG_GLOBAL_Z_FORCE**

FM_FORCE_MAGNITUDE

Local contact forces use the average local axis system of the parent segments meeting at the node. This is only an approximation.

XL_LOCAL_X_FORCE ***YL_LOCAL_Y_FORCE*** ***ZL_LOCAL_Z_FORCE***

Note: Contact force results are only available if a `.CTF` file is available, and forces are only available at nodes which form part of the topology of contact segments. For example forces at the "discrete nodes" which impact a surface, or nodes "spot-welded" to a surface, are not available.

12.6.3 Element data components averaged at nodes by D3PLOT

Where nodes are attached to underlying elements for which averaging at nodes is meaningful, (solids, shells, thick shells and contact segments), D3PLOT will report averaged values for scalar output.

The averaging logic is the same as that required to produce contour plots with one important exception: contouring a facet provides a "parent" element for the node, so that discontinuities between elements meeting at the node can be resolved by reference to the parent. However averaged data at nodes for scalar output cannot have such a "parent", and some logical errors can occur: for example averaging data where a node is common to elements of two different types.

Averaged STRESS data available at nodes:

(Solids, shells, thick shells)

The basic tensor components:

X_DIRECT_STRESS **XY_SHEAR_STRESS**

Y_DIRECT_STRESS **YZ_SHEAR_STRESS**

Z_DIRECT_STRESS **ZX_SHEAR_STRESS**

$$\begin{bmatrix} \sigma_x & & \\ \tau_{xy} & \sigma_y & \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix}$$

And the stresses derived from these by D3PLOT:

MAX_PRINC_STRESS ***MAX_DEV_PRINC_STRESS*** ***VON_MISES_STRESS***

MID_PRINC_STRESS ***MID_DEV_PRINC_STRESS*** ***MAX_SHEAR_STRESS***

MIN_PRINC_STRESS ***MIN_DEV_PRINC_STRESS*** ***PRESSURE***

Averaged STRAIN data available at nodes: (Solids, shells, thick shells)

The effective plastic strain:

STRAIN $[\epsilon_p]$

The basic strain tensor:

SX_DIRECT_STRAIN **SXY_SHEAR_STRAIN** $[\epsilon_x$ $]$

SY_DIRECT_STRAIN **SYZ_SHEAR_STRAINS** $[\epsilon_{xy}$ ϵ_y $]$

SZ_DIRECT_STRAIN **ZX_SHEAR_STRAIN**

And the strains derived from these by D3PLOT

SMAX_PRINC_STRAIN ***SVON_MISES_STRAIN***

SMID_PRINC_STRAIN ***SMAX_SHEAR_STRAIN***

SMIN_PRINC_STRAIN

The 1st 99 “extra” data components (if present)

SO1_SOLID_EXTRA_1 to **SO9_SOLID_EXTRA_99** (Solids only)

SH1_SHELL_EXTRA_1 to **SH9_SHELL_EXTRA_99** (Shell and thick shells only)

Further derived values

SR_STRAIN_RATE (Solids only)

RV_RELATIVE_VOLUME ***VOLUME*** (Solids and thick shells only)

Averaged FORCE and MOMENT resultants in shells available at nodes

Thin shell force and moment resultants (see section 12.8.7 for an explanation of these)

FX_NORMAL_FORCE **MX_BENDING_MOMENT** **QXZ_SHEAR_FORCE**

FY_NORMAL_FORCE **MY_BENDING_MOMENT** **QYZ_SHEAR_FORCE**

FXY_SHEAR_FORCE **MXY_BENDING_MOMENT**

And the stresses derived by D3PLOT from these

XA_AXIAL_ONLY **XB_BENDING_ONLY** **XO_OUTER_FIBRE**

YA_AXIAL_ONLY **YB_BENDING_ONLY** **YO_OUTER_FIBRE**

XYS_SHEAR_ONLY **XYO_OUTER_FIBRE**

Averaged thin shell miscellaneous components available at nodes:

INTERNAL_ENERGY_DENSITY

AS_AREA_OF_SHELL **THICKNESS**

Averaged contact segment STRESS components available at nodes:

Contact "stresses" in segments

CN_CONTACT_NORMAL **CT_CONTACT_TANGENTIAL**

COX_CONTACT_X **COY_CONTACT_Y**

12.6.4 Geometric data components available for output at nodes

These may be listed using **WRITE**, but not plotted in any way.

Topological data:

EN_ELEMENTS_AT_NODE (Lists elements attached to node)

Next section

12.7 Solid element data components.

12.7.1 The effect of Solid shape, element formulation and #integration points on output.

Solid elements in LS-DYNA may be 8 noded bricks, 6 noded wedges, or 4 or 10 noded tetrahedra. Their formulation may be single-point integration, 8 point integration, or reduced 20 noded.

Regardless of their shape, number of nodes or element formulation solid elements write out one of the following, depending on the variable <**NINTSLD**> on the ***DATABASE_EXTENT_BINARY** card.

NINTSLD = 1 (or 0) (The default case)	One set of results at the element centre. For those with more than one integration point this set of values is an average.
NINTSLD = 8	8 sets of results arranged at the 2 x 2 x 2 integration point locations. The effect of writing 8 integration points worth of data for elements with only a single integration point is undefined, and this option should only be used with fully integrated solids.
Versions of LS-DYNA prior to ls970 do not have this option, and only single integration point ouput is available from them.	

D3PLOT supports output at multiple integration points to a limited degree, although in most contexts only the value at the first integration point is used.

12.7.2 The results available for solid elements

Solid elements write the results to the **.PTF** file. The following tables show the raw data components, and those derived by D3PLOT.

Symmetric stress tensor:

$$\begin{bmatrix} \sigma_x & & \\ \tau_{xy} & \sigma_y & \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix}$$

X_DIRECT_STRESS **XY_SHEAR_STRESS**
YZ_SHEAR_STRESS
Y_DIRECT_STRESS
ZX_SHEAR_STRESS
Z_DIRECT_STRESS

This is written in the global cartesian coordinate system. It is always output, regardless of any of the switchable settings on the ***DATABASE_EXTENT_BINARY** control card. (See Section 9.2.1.3)

The stress components that can be derived this by D3PLOT are:

MAX_PRINC_STRESS	***MAX_DEV_PRINC_STRESS***	***VON_MISES_STRESS***
MID_PRINC_STRESS	***MID_DEV_PRINC_STRESS***	***MAX_SHEAR_STRESS***
MIN_PRINC_STRESS	***MIN_DEV_PRINC_STRESS***	***PRESSURE***

Effective plastic strain.

$$[\epsilon_p]$$

PLASTIC_STRAIN

The effective plastic strain is always output, as above. It has no intrinsic direction.

“Extra” variables for solids

If **NEIPH** has been defined on the ***DATABASE_EXTENT_BINARY** control card then **<neiph>** "extra" variables will be written. D3PLOT will accept any number of these, but will only process the first 99. These have the names:

SO1_SOLID_EXTRA_1 to **SO9_SOLID_EXTRA_99**

They are treated as separate scalar values of unknown type which may be contoured and written out, but not processed in any way.

Directional strain tensor for solids

$$[\epsilon_x \quad]$$

If **STRFLG** has been set on the ***DATABASE_EXTENT_BINARY** control card the symmetric strain tensor for solids will be written out. This is always oriented in the global cartesian coordinate system.

$$[\epsilon_x \quad]$$

SX_DIRECT_STRAIN **SXY_SHEAR_STRAIN**

SY_DIRECT_STRAIN **SYZ_SHEAR_STRAIN**

SZ_DIRECT_STRAIN **SZX_SHEAR_STRAIN**

$$[\epsilon_{xz} \quad \epsilon_{yz} \quad \epsilon_z]$$

And the strains derived from these by D3PLOT

SMAX_PRINC_STRAIN ***SVON_MISES_STRAIN***

SMID_PRINC_STRAIN ***SMAX_SHEAR_STRAIN***

SMIN_PRINC_STRAIN

Components derived geometrically by D3PLOT

Strain rate is calculated from the nodal displacements and velocities:

SR_STRAIN_RATE

Volume and relative volume ($Vol_{CURRENT}$ / $Vol_{Original}$) are calculated from nodal coordinates:

RV_RELATIVE_VOLUME ***VOLUME***

Geometric components

These components are extracted from the topology, and can be output in **WRITE**:

MN_MATERIAL_NUMBER **LN_LIST_OF_NODES** **FE_FACING_ELEMENTS**

12.7.3 Averaged nodal components for solids.

All the nodal results from the **.PTF** file listed in Section 9.6.1 are available for contouring on solid elements (in 2D/3D plotting mode). They can also be extracted as averaged element data for use in **WRITE** and **XY_PLOT**.

Nodal contact force components from the **.CTF** file cannot be processed on solid elements.

12.7.4 Transforming directional solid results to the element local system

It is possible to transform stress and strain tensor results from the global to the local element coordinate system using the **FRAME_OF_REFERENCE** options.

The local element axes, **[X', Y', Z']** for this purpose, are calculated as follows. If we adopt the notation that the vector from node 1 to node 2 is **N1N2**:

X' = **N1N2** approximately: the true **X'** vector is recomputed below

Z' = **N1N3 x N2N4** (Where **x** is a vector cross-product)

Y' = **Z' x X'**

X' = **Y' x Z'** This final transformation is required to correct for any warping

The formulae above are simplified for clarity. In 8 noded hexahedra the average of the bottom (N1N2N3N4) and top (N5N6N7N8) faces is used to determine a "middle" face; and for 6 noded wedges a similar averaging process is used. For tetrahedra Z' is obtained from N1N2 x N1N3.

12.7.5 Solid element results from thermal-only (TOPAZ3D) analyses

When a thermal-only analysis is run solid elements only are used.

However these do not write any results: the only output from such analyses is temperatures and flux vectors (in place of velocities) at nodes.

Thermal loading for structural models: the *LOAD_THERMAL_TOPAZ keyword.

It is possible to apply thermal loading to a structural model in the form of temperatures at nodes, varying over time, by using a "topaz" input file combined with the *LOAD_THERMAL_TOPAZ keyword. However this has to be a "pure" thermal file, formatted exactly as if it came from an old (1980s or 1990s) version of Topaz3d, and this is not the same as the output file that contemporary LS-DYNA generates from a thermal only analysis.

D3PLOT has an undocumented capability to generate "pure" Topaz files from contemporary "thermal only" output - please contact Oasys if you need to use this feature.

12.7.6 Solid element results from combined thermal and structural analyses

When a combined thermal/structural analysis is run the results for solids are exactly the same as those in a purely structural analysis.

12.7.7 Solid element results from an implicit NIKE3D analysis.

D3PLOT provides limited support for standalone NIKE3D (**N3PLOT**) files. Solid elements in these files report results at 8 integration points, and D3PLOT will process this, but this capability is still under development. Users wishing to use NIKE3D, either standalone or as embedded in LS-DYNA, should contact Oasys first for advice.

12.7.8 User-defined Solid data components.

D3PLOT 9.3 permits you to create an unlimited number of user-defined data components for solids. These may be either scalar or tensor:

- Scalar components are just treated as numerical values with no known properties or orientation.
- Tensor components are assumed to be in the global system when created, and are subject to Frame of Reference transformation as described in 12.7.4 above in exactly the same way as stresses and strains read from the database. The same derived components (principal, max shear, von Mises, etc) are also available.

Next section

12.8 Thin Shell element results.

12.8.1 Effect of shell shape and formulation on output

Thin shells in LS-DYNA may be quadrilateral or triangular; they may have a single integration point on plan, or be fully integrated; they may have from 1 to <n> integration points through their thickness; they may be linear (3 or 4 noded) or parabolic (6 or 8 noded).

Regardless of any of these permutations:

- ***On plan:*** ***shells only report results at their centre;***
- ***In elevation:*** ***shells only report results at*** **<maxint>** ***points through their thickness.***

(**<maxint>** is on the ***DATABASE_EXTENT_BINARY** card, and is explained in Section 12.2.1.3)

Sounds simple? Well it isn't! There is lots of scope for misunderstanding thin shell results, and if you have doubts please read this section. If you are still confused after that please contact Oasys for help and advice.

12.8.2 Description of shell output

Thin shells write out a large number of data values, and it is important that you understand exactly what they are. The greatest sources of confusion are the coordinate systems used and the relationship between integration points and data.

12.8.2.1 Thin shell coordinate systems

Global cartesian and element local.

Results are written in both of these systems depending on their nature. The local element axes, **[X',Y',Z']**, are calculated as follows. If we adopt the notation that the vector from node 1 to node 2 is **N1N2**:

X' = N1N2 approximately: the final **X'** vector recomputed below.

Z' = N1N3 x N2N4 (Where **x** is a vector cross-product)

Y' = Z' x X'

X' = **Y' x Z'** This final transformation is required to correct for any warping

Some components are written in the global cartesian system, and D3PLOT can transform these results to the local element system if required with the **FRAME_OF_REFERENCE** options. Examples are the stress and strain tensor values.

Other components are written in the element local system. Examples are the force and moment resultants.

We will revisit this topic later.

12.8.2.2 Thin shell integration points

From 1 to n through the thickness.

The majority of shell element formulations in LS-DYNA have a single integration point on plan, with from 1 to <n> integration points through their thickness. The "fully integrated" formulations have 2 x 2 integration points on plan, and again from 1 to <n> points through the thickness. For more information about shell formulations refer to the LS-DYNA theory manual.

For the purposes of this discussion we can ignore the number of integration points on plan, since they do not affect the output: thin shells only ever write results at the element centre as viewed on plan. What we are concerned with here is the number through the thickness, (as viewed in elevation) and we must consider two cases:

Case 1: MAXINT on the *DATABASE_EXTENT_BINARY card is set to the default of three.

In this situation, ***regardless of the number of integration points through the thickness used in the shell element formulation(s***), stress tensor and plastic strain output for all shell elements is written at three "surfaces":

"Top" : The outermost integration point

"Middle" : The neutral axis

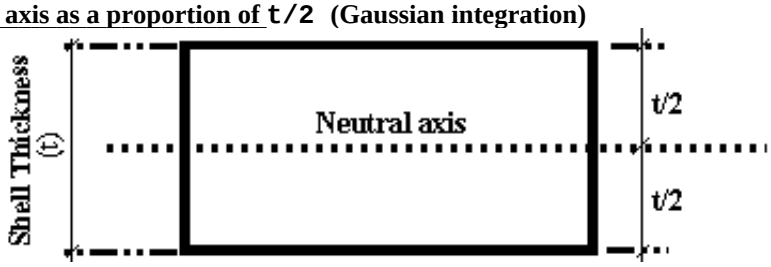
"Bottom" : The innermost integration point

The outer integration point is located on the +ve local Z' of the neutral axis, the inner on the -ve Z' side. (Membrane elements with a single integration point write the same value to all three surfaces.)

The important thing to note here is that, unlike linear-elastic codes, LS-DYNA reports "top and bottom surface" stresses at the outer and inner integration points, and ***not at the element outer fibres.***

Assuming the default Gaussian integration scheme is used the location of the outermost integration points is given in the following table as a function of shell thickness/2 (**t/2**) for the range of 1 to 6 integration points through the thickness.

No of Points	**Distance from neutral axis as a proportion of t/2 (Gaussian integration)**
1	0.0 (membrane)
2	0.577
3	0.775
4	0.861
5	0.906
6	0.932



Note:

Prior to LS-960	If the element formulation uses >5 integration points through its thickness then trapezoidal integration is used - even if Gaussian is specified..
From LS-960 onwards	The user manual claims that Gaussian integration is used for any number of points if specified.

It is possible to specify "user-defined" integration rules, which may locate integration points at different points through the thickness. For example Gauss-Lobatto quadrature will locate the outer and inner integration points on their respective outer fibres, but there is a speed penalty to pay for this.

In addition, in many analyses the section will be largely plastic, and the exact values of stress will be somewhat academic. Generally highly nonlinear problems are more concerned with plasticity and consequent energy absorbtion. However if you are in, or close to, the linear elastic region, and/or exact stress levels are important, you should consider using more integration points in order to extract results more precisely.

Case 2: "Layer" output with MAXINT set to some value other than the default of 3

In this situation LS-DYNA writes out the stress tensor and plastic strain values for the first **<maxint>** integration points in the shell, starting at the innermost (bottom) one. You would generally only use this for composites analysis where results in all layers are significant.

(1) LS-DYNA no longer calculates the "middle" neutral axis values for you.

If you have an odd number of integration points D3PLOT assumes that the middle one is the neutral axis value, if you have an even number of integration points D3PLOT averages the central pair to produce an approximate "middle" value.

(2) There is no check that **MAXINT** is equal to #integration points in a shell.

If **MAXINT** is less than the number of integration points in a shell you will be missing results; if greater the extra results will probably be junk. The true number of integration points in a shell is not written to the **.PTF** file, so D3PLOT cannot check this for you.

(3) There is a special case when **MAXINT** = 5.

If, and only if, MAXINT = 5 then results are written out in the integration point order: #3, #1, #2, #4, #5. There is no obvious reason for LS-DYNA doing this, and it is not documented anywhere, but a simple shell bending test model will reveal it to be true.

In this situation D3PLOT will still permit you to select "top", "middle" and "bottom" surfaces as before (with the "middle" surface being defined as in (1) or (3) above). But it will also permit you to define a Layer number instead in the range 1 to **<maxint>**. An example of this is shown in Section 4.2.4.2.

If you choose it this output option it is your responsibility to ensure that you interpret your results correctly.

If a ZTF file is not present D3PLOT has no means of knowing how many integration points a given element has actually specified on its section definition; nor whether it uses Gaussian, Lobatto or user-defined rules.

It is up to you to interpret your results correctly.

12.8.2.3 Cases where stress and strain tensor output are in the local coordinate system

Flag **CMPFLG** on the ***DATABASE_EXTENT_BINARY** card can be set to cause composite material stress and strain tensor output to be in the element (or material) local coordinate system. There is no way to detect this from the the **.PTF** file so, if you use this option, D3PLOT will assume that your results are still in the global system. Therefore it will be ***your responsibility to ensure that you interpret your results correctly.***

12.8.3 The results available for thin shell elements

Thin shell elements write the results to the **.PTF** file. The following tables show the raw data components, and those derived by D3PLOT.

Symmetric stress tensor:

$$\begin{bmatrix} \sigma_x & & \\ \tau_{xy} & \sigma_y & \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix}$$

X_DIRECT_STRESS
Y_DIRECT_STRESS
Z_DIRECT_STRESS

XY_SHEAR_STRESS
YZ_SHEAR_STRESS
ZX_SHEAR_STRESS

This is assumed to be written in the global cartesian coordinate system at **<maxint>** surfaces. Its output is on by default, but may be switched off using the **SIGFLG** parameter on the ***DATABASE_EXTENT_BINARY** control card. (See Section 9.2.1.3)

The stress components that can be derived this by D3PLOT are:

MAX_PRINC_STRESS	**MAX_DEV_PRINC_STRESS**	**VON_MISES_STRESS**
MID_PRINC_STRESS	**MID_DEV_PRINC_STRESS**	**MAX_SHEAR_STRESS**
MIN_PRINC_STRESS	**MIN_DEV_PRINC_STRESS**	**PRESSURE**

Effective plastic strain.

$$[\epsilon_p]$$

STRAIN

The effective plastic strain is written at the same **<maxint>** surfaces as the stress tensor. It has no intrinsic direction. By default it is switched **on**, but may be turned **off** using the **EPSFLG** parameter on the ***DATABASE_EXTENT_BINARY** control card.

"Extra" variables for thin shells

If **NEIPS** has been defined on the ***DATABASE_EXTENT_BINARY** control card then **<neips>** "extra" variables will be written at each of **<maxint>** surfaces (as for the stress tensor above). D3PLOT will accept any number of these, but will only process the first 99. These have the names:

SH1_SHELL_EXTRA_1 to **SH9_SHELL_EXTRA_99**

They are treated as separate scalar values of unknown type which may be contoured and written out, but not processed in any way.

Directional strain tensor for thin shells

$$\begin{bmatrix} \epsilon_x & & \\ \epsilon_{xy} & \epsilon_y & \\ \epsilon_{xz} & \epsilon_{yz} & \epsilon_z \end{bmatrix}$$

If **STRFLG** has been set on the ***DATABASE_EXTENT_BINARY** control card the symmetric strain tensor for thin shells will be written out at the innermost and outermost integration points only, regardless of the value of **<maxint>**. This is assumed to be oriented in the global cartesian coordinate system.

SX_DIRECT_STRAIN **SXY_SHEAR_STRAIN**

SY_DIRECT_STRAIN **SYZ_SHEAR_STRAIN**

SZ_DIRECT_STRAIN **SZX_SHEAR_STRAIN**

And the strains derived from these by D3PLOT

SMAX_PRINC_STRAIN ***SVON_MISES_STRAIN***

SMID_PRINC_STRAIN ***SMAX_SHEAR_STRAIN***

SMIN_PRINC_STRAIN

Shell force and moment resultants

Shell force and moment resultants are written as **<Force/unit width>** and **<moment/unit width>** in the element local cordinate system. By default these are **on**, but they can be switched **off** with the **RLTFLG** flag on the ***DATABASE_EXTENT_ BINARY** control card. (These components are explained in section 12.8.7)

FX_NORMAL_FORCE **MX_BENDING_MOMENT** **QXZ_SHEAR_FORCE**

FY_NORMAL_FORCE **MY_BENDING_MOMENT** **QYZ_SHEAR_FORCE**

FXY_SHEAR_FORCE **MXY_BENDING_MOMENT**

Stresses derived by D3PLOT from the force & moment resultants

XA_AXIAL_ONLY ***XB_BENDING_ONLY*** ***XO_OUTER_FIBRE***

YA_AXIAL_ONLY ***YB_BENDING_ONLY*** ***YO_OUTER_FIBRE***

XYS_SHEAR_ONLY ***XYO_OUTER_FIBRE***

Thickness and Internal energy density components

By default these components are written. They can be turned **off** with the **ENGFLG** flag on the ***DATABASE_EXTENT_BINARY** control card.

INTERNAL_ENERGY_DENSITY

THICKNESS

Components derived geometrically by D3PLOT

Shell area is calculated from the nodal coordinates:

AS_AREA_OF_SHELL ***ON_OUTWARD_NORMAL***

Geometric components

These components are extracted from the topology, and can be output in **WRITE**:

MN_MATERIAL_NUMBER LN_LIST_OF_NODES

12.8.4 Averaged nodal data components for thin shells.

All the nodal results from the **.PTF** file listed in Section 12.6.1 are available for contouring on the shell elements (in 2D/3D plotting mode). They can also be extracted as averaged element data for use in **WRITE** and **XY_PLOT**.

Nodal contact force components from the **.CTF** file **cannot** be processed on thin shell elements.

Next section

12.8.5 Out of plane (Z') stress tensor components

Thin shells in LS-DYNA do not calculate the element through-thickness ($\sigma_{Z'}$) direct stress which will always be zero, although the through thickness shear terms ($\tau_{YZ'}$ and $\tau_{XZ'}$) are calculated. Thus five of the six terms of the local stress tensor are used, giving:

$$\begin{bmatrix} \sigma_{x'} & & \\ \tau_{xy'} & \sigma_{y'} & \\ \tau_{xz'} & \tau_{yz'} & 0 \end{bmatrix}$$

This means that principal stresses will not necessarily lie in the plane of the shell.

12.8.6 Out of plane (Z') strain tensor components

Thin shells do not integrate the through thickness strain ($\varepsilon_{Z'}$), but this value may nevertheless be reported as non-zero. This is because the shell thickness may change, leading to a strain: its stiffness in this direction is based on volume preservation, but there is no stress associated with this.

Therefore principal strains also may not be confined to the plane of the element.

12.8.7 FX_ etc Explanation of shell force and moment resultants.

These are integrals, computed by LS-DYNA, of the stresses in the shell local system to create force and moment values per unit width in the element local system.

The sign convention is taken from "Theory of Elastic Stability" Timoshenko and Gere, and the equations to derive the resultants from the local stress values are given below. (**t** is the shell thickness.)

$$F_X = \int_{-t/2}^{+t/2} \sigma'_X . dt \qquad F_Y = \int_{-t/2}^{+t/2} \sigma'_Y . dt \qquad F_{XY} = \int_{-t/2}^{+t/2} \tau'_{XY} . dt$$

$$M_X = \int_{-t/2}^{+t/2} t . \sigma'_x . dt \qquad M_Y = \int_{-t/2}^{+t/2} t . \sigma'_Y . dt \qquad M_{XY} = \int_{-t/2}^{+t/2} t . \tau'_{XY} . dt$$

$$Q_{XX} = \int_{-t/2}^{+t/2} \tau'_{XZ} . dt \qquad Q_{YY} = \int_{-t/2}^{+t/2} \tau'_{YZ} . dt$$

These are the forces and moments per unit width integrated over the element thickness. They are written in the element local axis system.

12.8.8 XA_ etc Stresses in thin shells derived from force and moment resultants

The force/moment resultants are converted to local axial and bending stresses as follows:

Axial stress	=	F/t
Bending stress	=	$6M/t^2$
Outer fibre stress	=	$F/t \pm 6M/t^2$

Where: **F** = force / unit width

M = moment / unit width

t = section thickness

Note that the bending component here assumes a ***linear elastic stress*** distribution (ie stress = **My/I**) so "bending" and "outer fibre" stresses will be ***incorrect*** in a plastic section.

12.8.9 Summary of coordinate systems and default locations of thin shell results

This table shows the coordinate system and location of each category of data written by LS-DYNA, and also the same information for results derived from these by D3PLOT. It assumes the default of **MAXINT** = 3.

TYPE OF DATA	COORDINATE SYSTEM	ELEMENT LOCATION
Basic global stress tensor	Global cartesian axes (Can transform to local)	Inner integration point Neutral axis Outer integration point
Stresses derived from global tensor	n/a	Inner integration point Neutral axis Outer integration point
Plastic strain	n/a	Inner integration point Neutral axis Outer integration point
Force/moment resultants	Element local axes	Whole element
Stresses derived from force/moment resultants	Element local axes	Outer fibre Neutral axis
Directional strain tensor	Global cartesian axes (Can transform to local)	Inner integration point Outer integration point
Extra data at integration points	Undefined	Inner integration point Neutral axis Outer integration point
Additional components	n/a	Whole element

Coordinate system and default locations for thin shell element data

Notes on this table:

- The entry "n/a" in the coordinate system column means that the data does not have a fixed direction. For example **VON_MISES** or **PRINCIPAL** stresses. The words "can transform to local" means that D3PLOT global to local transformation is available.
- The entry "whole element" in the element location column means that the data is for the element as a whole. For example **FXX_AXIAL_FORCE** or **THICKNESS** components.

- The "outer fibre" stresses calculated from force/moment resultants assume a linear elastic stress distribution the element. This may not always be the case.

12.8.10 User-defined shell components.

D3PLOT 9.3 permits you to create an unlimited number of user-defined data components for shells. These may be either scalar or tensor:

- Scalar components are just treated as numerical values with no known properties or orientation.
- Tensor components are assumed to be in the global system when created, and are subject to Frame of Reference transformation in exactly the same way as stresses and strains read from the database. The calculation of the element local system is explained in section 12.8.2.1 above. The same derived components (principal, max shear, von Mises, etc) are also available.

12.9 Thick shell element results

Thick shells in LS-DYNA look like bricks, but their internal formulation is more that of a shell. For example they use very similar integration schemes and, like thin shells, do not integrate through thickness (Z') stress. (However through thickness strain is computed from volume conservation.)

12.9.1 Effect of shell shape and formulation on output

Thick shells in LS-DYNA may be hexahedra or extruded triangles. They may have a single integration point on plan, or be fully integrated. They may have from 1 to 5 integration points through their thickness.

However, like thin shells:

On plan: ***thick shells only report results at their centre;***

In elevation: ***thick shells only report results at*** **<maxint>** ***points through their thickness.***

(**<maxint>**is on the ***DATABASE_EXTENT_BINARY** card, and is explained in Section 9.2.1.3)

In most respects thick shells are like thin shells, and you are encouraged to read Section 9.8.2 which describes some of the pitfalls of using them. This will not be repeated here.

12.9.2 The results available for thick shell elements

Thick shell elements write the results to the **.PTF** file. The following tables show the raw data components, and those derived by D3PLOT.

Symmetric stress tensor:

X_DIRECT_STRESS **XY_SHEAR_STRESS**
Y_DIRECT_STRESS **YZ_SHEAR_STRESS**
Z_DIRECT_STRESS **ZX_SHEAR_STRESS**

$$\begin{bmatrix} \sigma_x & & \\ \tau_{xy} & \sigma_y & \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix}$$

This is assumed to be written in the global cartesian coordinate system at **<maxint>** surfaces. Its output is **on** by default, but may be switched **off** using the **SIGFLG** parameter on the ***DATABASE_EXTENT_BINARY** control card. (See Section 12.2.1.3)

The stress components that can be derived this by D3PLOT are:

MAX_PRINC_STRESS **MAX_DEV_PRINC_STRESS** **VON_MISES_STRESS**

MID_PRINC_STRESS **MID_DEV_PRINC_STRESS** **MAX_SHEAR_STRESS**

MIN_PRINC_STRESS **MIN_DEV_PRINC_STRESS** **PRESSURE**

Effective plastic strain. $[\epsilon_p]$

STRAIN

The effective plastic strain is written at the same **<maxint>** surfaces as the stress tensor. It has no intrinsic direction. By default it is switched on, but may be turned off using the **EPSFLG** parameter on the ***DATABASE_EXTENT_BINARY** control card.

"Extra" variables for thick shells

If **NEIPS** has been defined on the ***DATABASE_EXTENT_BINARY** control card then **<neips>** "extra" variables will be written at each of **<maxint>** surfaces (as for the stress tensor above). D3PLOT will accept any number of these, but will only process the first 99. These have the names:

SH1_SHELL_EXTRA_1 to **SH9_SHELL_EXTRA_99**

They are treated as separate scalar values of unknown type which may be contoured and written out, but not processed in any way.

Directional strain tensor for thick shells

$$\begin{bmatrix} \epsilon_x & & \\ \epsilon_{xy} & \epsilon_y & \\ \epsilon_{xz} & \epsilon_{yz} & \epsilon_z \end{bmatrix}$$

If **STRFLG** has been set on the ***DATABASE_EXTENT_BINARY** control card the symmetric strain tensor for thick shells will be written out ***at the innermost and outermost integration points only, regardless of the value of*** **<maxint>**. This is assumed to be oriented in the global cartesian coordinate system.

SX_DIRECT_STRAIN **SXY_SHEAR_STRAIN**

SY_DIRECT_STRAIN **SYZ_SHEAR_STRAIN**

SZ_DIRECT_STRAIN **SZX_SHEAR_STRAIN**

And the strains derived from these by D3PLOT

SMAX_PRINC_STRAIN ***SVON_MISES_STRAIN***

SMID_PRINC_STRAIN ***SMAX_SHEAR_STRAIN***

SMIN_PRINC_STRAIN

Components derived geometrically by D3PLOT

Thick shell volume and relative are calculated from the nodal coordinates:

VOLUME ***RV_RELATIVE_VOLUME***

Geometric components

These components are extracted from the topology, and can be output in **WRITE**:

MN_MATERIAL_NUMBER **LN_LIST_OF_NODES** **FE_FACING_ELEMENTS**

12.9.3 Averaged nodal data components for thick shells.

All the nodal results from the **.PTF** file listed in Section 12.6.1 are available for contouring on thick shell elements (in 2D/3D plotting mode). They can also be extracted as averaged element data for use in **WRITE** and **XY_PLOT**.

Nodal contact force components from the **.CTF** file **cannot** be processed on thick shell elements.

12.9.4 Out of plane (Z') stress tensor components

Thick shells in LS-DYNA do not calculate the element through-thickness ($\sigma_{z'}$) direct stress which will always be zero, although the through thickness shear terms ($\tau_{YZ'}$ and $\tau_{XZ'}$) are calculated. This means that these elements should not be used where a full three-dimensional stress state is expected to develop.

Thus five of the six terms of the local stress tensor are used, giving:

$$\begin{bmatrix} \sigma_x & & \\ \tau_{xy} & \sigma_y & \\ \tau_{xz} & \tau_{yz} & 0 \end{bmatrix}$$

This means that principal stresses will not necessarily lie in the plane of the shell.

12.9.5 Out of plane (Z') strain tensor components

Thick shells do not **integrate** the through thickness strain ($\epsilon_{Z'}$), but this value may nevertheless be reported as non-zero. This is because the shell thickness may change, leading to a strain: its stiffness in this direction is based on volume preservation, but there is no stress associated with this.

Therefore principal strains also may not be confined to the plane of the element.

12.9.6 Frame of reference: computing the local coordinate system

The local coordinate system used for thick shells is computed in much the same way as for shells, except that the average of the bottom (N1N2N3N4) and top (N5N6N7N8) faces is used to produce a "middle" face. The formulae are, as with shells:

X' = **N1N2** approximately: the true **X'** vector is recomputed below

Z' = **N1N3 x N2N4** (Where **x** is a vector cross-product)

Y' = **Z' x X'**

X' = **Y' x Z'** This final transformation is required to correct for any warping

In the case of a 6 noded thick tria Z' is obtained from N1N2 x N1N3

12.9.7 User-defined Thick shell data components.

D3PLOT 9.3 permits you to create an unlimited number of user-defined data components for solids. These may be either scalar or tensor:

- Scalar components are just treated as numerical values with no known properties or orientation.
- Tensor components are assumed to be in the global system when created, and are subject to Frame of Reference

transformation as described in section 12.9.6 above in exactly the same way as stresses and strains read from the database. The same derived components (principal, max shear, von Mises, etc) are also available.

Next section

12.10 Beam element results

Beams in LS-DYNA are always linear, 2-noded elements. There are several different formulations but the two principal types are:

Resultant:	Used for standard sections, and special "plastic" resultant formulations.
Integrated:	Also used for standard sections, but also for user-defined arbitrary sections.

Note that due to a bug in LS-DYNA the output sign convention used for the two types above is inconsistent up to and including LS-DYNA release 970, this problem has been fixed from LS-DYNA 971 onwards. This issue is described in section 12.2.1.3 where controlling beam output is discussed. This has implications for beam plotting (section section 4.2.5) and when extracting cut section forces and moments through beam structures (see section 6.4.4).

12.10.1 “Basic” components for all beams.

All beams types generate a "basic" force and moment vector (even if some of them populate it entirely with zeros!). This is [**Fx, Fy, Fz, Mxx, Myy, Mzz**], where all results are written in the local axis system for beams. The D3PLOT components have the names:

FX_AXIAL_FORCE	**MXX_TORSIONAL_MOMENT**
FY_Y_SHEAR_FORCE	**MYY_BENDING_MOMENT**
FZ_Z_SHEAR FORCE	**MZZ_BENDING MOMENT**

The local axis system for beams is derived as follows:

If N1N2 is the vector from node 1 to node 2:

X’ = **N1N2**

Z’ = **N1N2 x N1N3** (Where x is the vector cross-product)

Y’ = **Z’ x X’**

The "third" node (N3) is the orientation node for the beam, serving only to define its local Y’ axis. It has no structural significance. If the representation of beam local axes during post-processing is important throughout an analysis you should consider defining separate "third" nodes for each beam element, and setting the <**nrefup**> field on the ***CONTROL_OUTPUT** card to update these nodes’ coordinates.

12.10.2 “Extra” components for Hughes-Liu beams

If the **BEAMIP** flag on the ***DATABASE_EXTENT_BINARY** card is used "extra" data for **<beamip>** integration points in Hughes-Liu beams will be output. The 5 components for each point are:

XX_AXIAL_STRESS	**SP_PLASTIC_STRAIN**
YY_SHEAR_STRESS	**SAX_AXIAL_STRAIN**
ZZ_SHEAR_STRESS	

These values are calculated by LS-DYNA, and output in the relevant beam local axes.

12.10.3 “Extra” components for Belytschko-Schwer beams

If **BEAMIP** on the ***DATABASE_EXTENT_BINARY** card is set to 3 or more the following extra data components will be written for all Belytschko-Schwer beams:

MY1_Y_BENDING_MOM_END_1 **MZ1_Z_BENDING_MOM_END_1**

MY2_Y_BENDING_MOM_END_2 **MZ2_Z_BENDING_MOM_END_2**

MYD_Y_MOM_DISTRIBUTION **MZD_Z_MOM_DISTRIBUTION**

MMD_MOM_MAG_DISTRIBUTION

And if the Belytschko-Schwer beams use a resultant material formulation:

RXX_PLASTIC_TORS_ROTN

RY1_Y_PLASTIC_ROT_END_1 **RZ1_Z_PLASTIC_ROT_END_1**

RY2_Y_PLASTIC_ROT_END_2 **RZ2_Z_PLASTIC_ROT_END_2**

RYD_Y_ROT_DISTRIBUTION **RZD_Z_MOM_DISTRIBUTION**

PE1_PLASTIC_ENERGY_END_1 **RMD_ROT_MAG_DISTRIBUTION**

PE2_PLASTIC_ENERGY_END_2 **PED_PLASTIC_ENERGY_DIST**

SAX_TOTAL_AXIAL_STRAIN **EAX_AXIAL_ENERGY**

IE_INTERNAL_ENERGY

BED_BENDING_ENERGY_DENS = **`(PE1 + PE2) / Length`**

AED_AXIAL_ENERGY_DENS = **`EAX / Length`**

IED_INTERNAL_ENERGY_DEN = **`IE / Length`**

12.10.4 Notes on beam data.

(1) Hughes-Liu beams locate their integration point(s) at mid-span, and have a constant shear force and moment along their length.

Belytschko-Schwer beams calculate the moment variation along the beam, so may have different Myy and Mzz terms at ends one and two. This presents a problem when only the basic force and moment vector is written since only one Myy and one Mzz term are output. These are in fact the values at node 1. So if you have a cantilever fixed at end 2, with a point load at end 1, you will not see any moment in it (although it will behave correctly).

The best solution to this problem is to write the "extra" data since, as is shown above, separate end 1 and end 2 moments are then written to file.

(2) At present there is no way to tell from the database whether the "extra" data is for Belyschko-Schwer or Hughes-Liu beams. So both sets of options may be extracted from the same dataset: ***it is your responsibility to interpret your data correctly.***

12.11 Contact segment results

This section describes the results available for individual contact segments, as distinct from those for surfaces as a whole as described in Section 4.2.6.

Contact segment results are written to the **`.CTF`** file, so if this file is missing it will not be possible to visualise or process contact surface results.

12.11.1 What are contact segments?

They are not really elements, although it is convenient to treat them as such within D3PLOT. They are 3 or 4 noded areas over which contact is calculated, and which must lie on "real" structural elements underneath.

Contact forces are calculated at nodes, and then averaged over the area of their connected segments to give contact

"stress": really these value are "pressure" not "stress" (although the units are the same).

Users should be aware that contact forces in LS-DYNA are calculated from the repulsion forces required to stop nodes penetrating surfaces, and that this is an inherently noisy process since penetrations - and hence forces - tend to oscillate. Therefore contact forces on individual segments at a given state should be treated as a snapshot of a dynamic process, and not necessarily a good indication of the mean contact force averaged over a longer time period.

12.11.2 Components written by LS-DYNA for contact segments.

Contact segment "stress" values written by LS-DYNA at segments:

CN_CONTACT_NORMAL	Stress normal to surface;
CT_CONTACT_TANGENTIAL	Resultant stress in plane of surface;
CX_CONTACT_X	In-plane local X stress;
CY_CONTACT_Y	In-plane local Y stress;

Contact forces written by LS-DYNA at nodes, averaged by D3PLOT over segments:

XG_GLOBAL_X_FORCE
YG_GLOBAL_Y_FORCE
ZG_GLOBAL_Z_FORCE

XL_LOCAL_X_FORCE
YL_LOCAL_Y_FORCE
ZL_LOCAL_Z_FORCE

FM_FORCE_MAGNITUDE

Local forces are transformed to reflect the orientation of a segment. But since the nodal forces on which they are based also have contributions from adjacent segments they should be regarded as approximate.

Versions of LS-DYNA from approximately ls960 (~2002) onwards may also write:

Contact gap at node N1
Contact gap at node N2
Contact gap at node N3
Contact gap at node N4

Surface energy density of segment

D3PLOT wil process files containing these data, but they are ignored and are not plottable.

12.11.3 Geometric components calculated by D3PLOT

ON_OUTWARD_NORMAL	Special geometric component to show segment orientation.

12.11.4 Results for whole surfaces

D3PLOT can also sum up results for contact surfaces, and the results can be viewed numerically in **WRITE** and **XY_DATA**. These values are the numerical sum of the relevant component for all segments in the surface.

When LS-DYNA writes contact surface results to the .CTF file it distinguishes between Master and Slave sides of each contact, therefore D3PLOT allows you to report results for contact surfaces as follows:

Master side	Results are computed from segments on the master side
Slave side	Results are computed from segments on the slave side

<table>
<tr>
<td>Surface as a whole where the summary results from master and slave sides are within 5% by magnitude.

(Master and slave sides should be equal in magnitude, and directional components opposite in sign.)</td>
<td>
<table>
<tr><td>Directional (eg X force)</td><td>(Master - slave) * 0.5</td></tr>
<tr><td>Magnitude values</td><td>(Master + slave) * 0.5</td></tr>
</table>
Therefore the sign of the output of directional values is that of the master side.
</td>
</tr>
<tr>
<td>Surface as a whole where summary results from master and slave sides differ by more than 5% by magnitude. (Typically the single-surface case, or "nodes to" types with no slave segments.)</td>
<td>Whichever side has the greater value by magnitude, with no sign change applied.</td>
</tr>
</table>

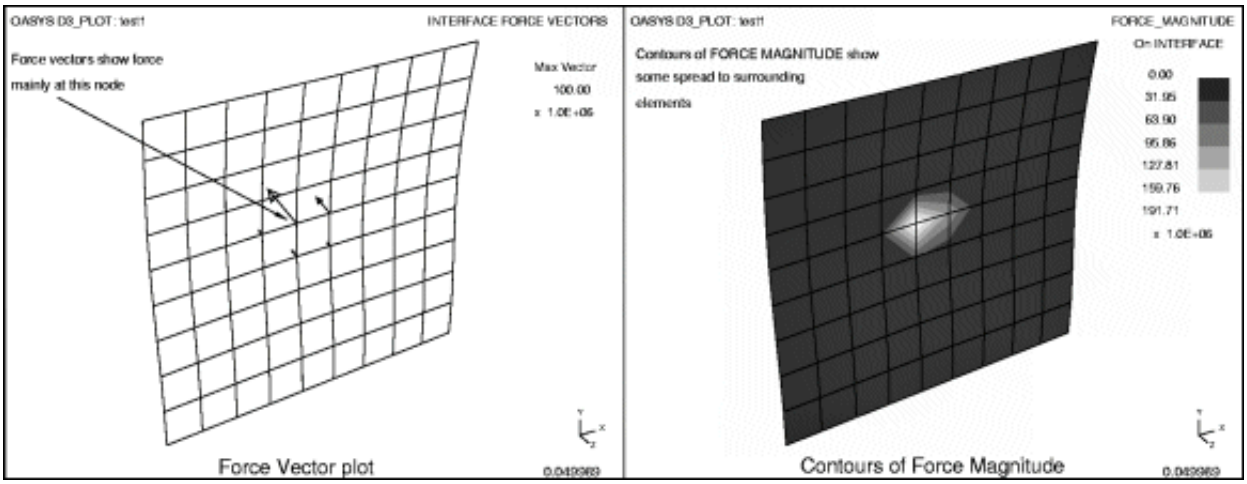
12.11.5 How LS-DYNA calculates and D3PLOT processes contact “stresses”

IMPORTANT: The "stresses" on interfaces will always be lower than the true values in the elements, and they will also be spread over a wider area.

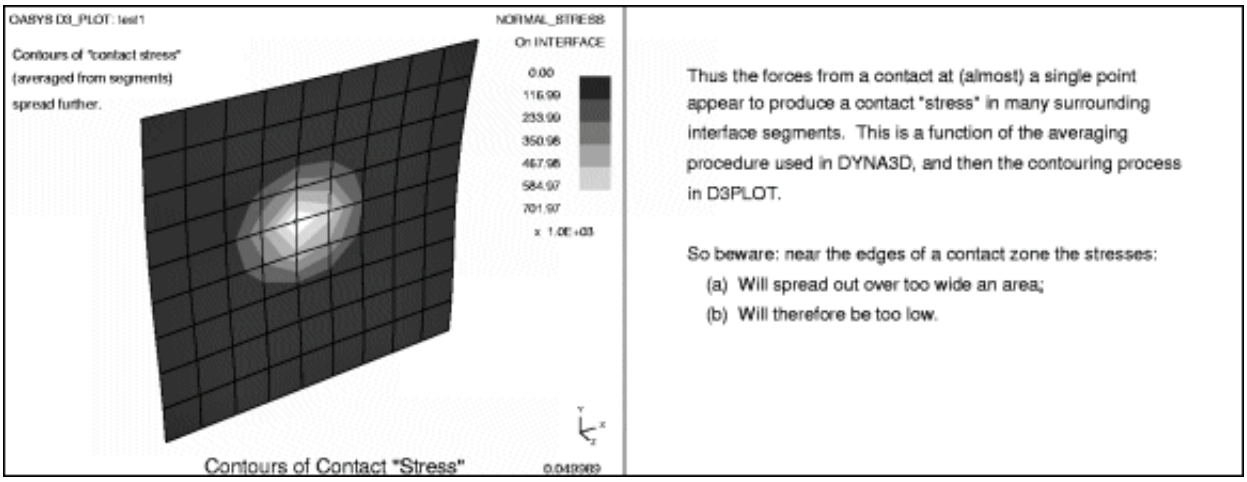
This is because of the way the penalty force and contouring algorithms work. Consider the following example (see the three figures below)

a) A single node (say on the master side) touches part of the (slave) surface. (Master and slave are inter-changeable in this example.)

b) The reaction forces on the slave side are distributed among the four nodes on the slave segment.



c) Contact stress is then computed over the sphere of influence of each node on the slave surface, and stress is assigned to each contact segment.


Contours of "contact stress"
(averaged from segments)
spread further.
NORMAL_STRESS
On INTERFACE
0.00
116.99
233.99
350.96
467.96
584.97
701.97
x 1.0E+03
Contours of Contact "Stress"
Thus the forces from a contact at (almost) a single point appear to produce a contact "stress" in many surrounding interface segments. This is a function of the averaging procedure used in DYNA3D, and then the contouring process in D3PLOT.
So beware: near the edges of a contact zone the stresses:
(a) Will spread out over too wide an area;
(b) Will therefore be too low.

d) Contouring which, in D3PLOT averages over elements, spreads the stress over a further region of contact segments.

Thus the contact at a single point has generated an apparent stress spread over twenty five elements - clearly this is not correct. To get round this you can:

- Plot "force" components (eg **FM_FORCE_MAGNITUDE**) which omit step (c) above, so force is spread only over nine segments in this example.
- Plot force vectors (**VECTOR** command) which will only show force arrows at the four nodes on the slave segment.

However in real examples the problem is less severe: the genuine contact is usually spread over several segments and it is only around the borders of the contact region that stresses spread out too far.

Nevertheless contact "stresses" should not be treated as more than approximate contact pressures, and in particular they must not be expected to be the same as (true) stress in the underlying elements.

Next section

12.12 Smooth Particle Hydrodynamic (SPH) Data components

From release 9.3 D3PLOT will process SPH elements if present in the database.

12.12.1 SPH membership of PARTs

SPH elements belong to PARTs in exactly teh same way as solids, shells and beams; so D3PLOT processes them "by part", and makes their part-based data components available in exactly the same way.

In addition it will be clear from the tables below that the stress and strain SPH database components are the same as solid and shell ones, so they are contoured alongside them in 2D3D plotting mode, fitting in neatly with PART membership.

12.12.2 SPH Data organisation

SPH elements can be thought of as having a single integration point at their centre, and all directional tensor components are written out in the global model system. The global/local frame of reference transformation has no effect on these results, and the current surface/integration point setting also does not affect them.

SPH elements can be deleted, as with other elements. D3PLOT detects this and removes deleted elements from the plot.

12.12.3 The results available for SPH elements

SPH elements write out the following block of 18 values per element to the **.PTF** file:

- Radius of influence (1 value, used to determine its size when drawn)
- Pressure (1 value)
- Stress tensor (6 values)
- Plastic strain (1 value)
- Density (1 value)
- Internal energy (1 value)
- Number of neighbours (1 value)
- Strain tensor (6 values)

Experience suggests that the above data components are written regardless of any flags on ***DATABASE_EXTENT_BINARY** or other control cards.

The following tables show the raw data component names, and also those derived by D3PLOT.

Symmetric stress tensor: $\begin{bmatrix} \sigma_x & & \\ \tau_{xy} & \sigma_y & \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix}$

X_DIRECT_STRESS
Y_DIRECT_STRESS
Z_DIRECT_STRESS
XY_SHEAR_STRESS
YZ_SHEAR_STRESS
ZX_SHEAR_STRESS

This is written in the global cartesian coordinate system.

The stress components that can be derived this by D3PLOT are:

MAX_PRINC_STRESS	***MAX_DEV_PRINC_STRESS***	***VON_MISES_STRESS***
MID_PRINC_STRESS	***MID_DEV_PRINC_STRESS***	***MAX_SHEAR_STRESS***
MIN_PRINC_STRESS	***MIN_DEV_PRINC_STRESS***	***PRESSURE***

Effective plastic strain. $[\epsilon_p]$

PLASTIC_STRAIN

The effective plastic strain is always output, as above. It has no intrinsic direction.

Directional strain tensor

$$\begin{bmatrix} \epsilon_x & & \\ & & \\ \epsilon_{xz} & \epsilon_{yz} & \epsilon_z \end{bmatrix}$$

Unlike other element types this strain tensor is written unconditionally, regardless of the value of **STRFLG** on the ***DATABASE_EXTENT_BINARY** card.

SX_DIRECT_STRAIN **SXY_SHEAR_STRAIN**

SY_DIRECT_STRAIN **SYZ_SHEAR_STRAIN**

SZ_DIRECT_STRAIN **SZX_SHEAR_STRAIN**

And the strains derived from these by D3PLOT

SMAX_PRINC_STRAIN ***SVON_MISES_STRAIN***

SMID_PRINC_STRAIN ***SMAX_SHEAR_STRAIN***

SMIN_PRINC_STRAIN

Further SPH-only components

RADIUS	**NUM_NEIGHBOURS**	It is not clear why PRESSURE is written separately, rather than being deduced from the stress tensor. D3PLOT uses this value rather than (Sx + Sy + Sz) / -3.0
VOLUME	**ENERGY**	
DENSITY	**PRESSURE**	

The following are derived from the above:

VOLUME (from radius) **MASS** (from **VOLUME** x **DENSITY**)

12.13 Airbag Particle (ABP) data components

From release 9.3 D3PLOT processes Airbag Particle elements.

These can be thought of as small spherical particles which are emitted from an inflator to apply pressure to an airbag, as an alternative to using a control volume. Each inflator contains one or more gas generators, which emit particles as the analysis progresses. This means that - in effect - particles are "born" when they first appear in the analysis, and while they don't "die" as such they may pass through the fabric of the bag (or through a vent hole) and cease to take an active part in the analysis. Therefore the number of particles is initially zero, and their quantity increases as they pop into existence as the analysis progresses.

The way particles act upon the fabric of the bag is effectively to make contact with it and hence apply force, mimicing the true behaviour of actual gas particles - albeit on a much larger scale.

Airbag particles are treated as elements within D3PLOT, and such data values are as contourable are displayed in the 2D3D plotting mode. However as will be clear from the tables below these particles act more like small rigid balls and don't have element data as such, so the main purpose in plotting them is to see how they are inflating the airbag and "contouring" their data is nearly meaningless.

At the time of writing (September 2008) the information available about this feature is limited, and the information below has been largely reverse-engineered from what is present in databases written from example analyses run in

LS-DYNA 971R4. It is likely that this feature will be developed further, and generate more information in the future.

12.13.1 ABP membership of "Airbags"

Within D3PLOT airbag particles belong to "Airbags" in exactly the same sense that solids, shells and beams belong to "Parts". (The "airbag" is the Control Volume definition in the input deck.) It is not possible to tell from the .ptf file what the fabric elements making up a bag are, so it cannot be drawn explicitly, however being able to select and process "by airbag" fits neatly into the logic of the programme, and makes it easy to select and control these elements.

"Per airbag" data can be plotted (all particles in a bag getting the same contour value), and can also be displayed numerically in **WRITE** and **XY_DATA**. At present the data components available at "Airbag" level are:

ANP_ABAG_NUM_PARTICLES the number of "live" particles in this bag

AVOL_AIRBAG_VOLUME the current bag volume

12.13.2 ABP data components

The following data components are written from LS-DYNA 971R4 and may be "contoured" in 2D3D plotting mode as data components. "Contouring" is a misnomer in this context, since each particle is independent and has a single value, however the display of this value can still be useful.

Current coordinates

CX_CURRENT_X_COORD **CY_CURRENT_Y_COORD** **CZ_CURRENT_Z_COORD**

Displacements (Derived from **<current> - <undeformed>**)

DX_X_DISPLACEMENT ***DY_Y_DISPLACEMENT*** ***DZ_Z_DISPLACEMENT***

DR_DISP_RESULTANT

Velocities

VX_X_VELOCITY **VY_Y_VELOCITY** **VZ_Z_VELOCITY**

VR_VEL_RESULTANT

Other

MASS

RADIUS

NS_DIST (Distance to nearest bag segment)

VOLUME (from **RADIUS**)

DENSITY (from **MASS** / **VOLUME**)

GAS_ID (The gas number)

LEAKAGE (0 inside bag, 1 escaped, 2 & 3 ??)

SPIN_ENERGY

TRANS_ENERGY

12.13.3 Nodes on ABP elements: VISFLG on *AIRBAG_PARTICLE card

The documentation on the ***AIRBAG_PARTICLE** card suggests that **VISFLG** must be turned on in order to see airbag particles. This is misleading: airbag particle data is written to the .ptf file regardless if present in the analysis, and this flag simply turns on the output of nodes coincident with these elements.

Since D3PLOT 9.3 onwards will visualise ABP elements without needing these nodes it is recommended that you turn the **VISFLG** off in order to reduce the output database size.

12.14 Data components for other entity types

LUMPED_MASSES, **SPRINGS**, **SEAT_BELTS** etc, **JOINTS**, **STONEWALLS**:

D3PLOT cannot currently extract time-dependent data for these element types, (except normal force on stonewalls), so only geometric data components are available for them. These data are extracted from the **.xtf** file if present, or from the **.ztf** file if not. If neither file is present then these items will not be processed.

12.14.1 **LUMPED-MASS** components

LN_LIST_OF_NODES **CE_CONNECTED_ELEMENTS** **MASS**

SUMMARY

12.14.2 **SPRING/DAMPER** components

MN_MATERIAL_NUMBER **LN_LIST_OF_NODES** **TYPE_OF_SPRING**

SUMMARY

In addition if the **BEAM** flag on the ***DATABASE_BINARY_D3PLOT** card may be used to write spring results into beam "slots" in the .ptf file as follows:

BEAM = 0 (default)	Extra beam elements are added to the .ptf file using the spring/damper topology, and Global [Fx, Fy, Fz, Fr] forces are written in the Fx, Fy, Fz, Myy data "slots" for beams
BEAM = 1	No extra beams are written
BEAM = 2	Extra beam elements are written as for the "0" case above, but only the resultant force is written to the Fx "slot".

If these extra beams are present in the database D3PLOT has no way of knowing whether they are genuine beams, or springs masquerading as beams. Therefore if you use this option treat your results with care.

12.14.3 **SEAT_BELT, RETRACTOR and SLIP_RING** components

MN_MATERIAL_NUMBER **LN_LIST_OF_NODES** **SUMMARY**

Since seatbelt elements are really discrete elements inside LS-DYNA the BEAM flag on *DATABASE_BINARY_D3PLOT will also result in beams being written out for seatbelt elements as described in 12.14.2 above.

12.14.4 **JOINT** components

LN_LIST_OF_NODES **TYPE_OF_JOINT** **STIFFNESS**

SUMMARY

12.14.5 **STONEWALL** components

NORMAL_FORCE **SUMMARY**

Notes:

- The **SUMMARY** components above are the most useful since they list all relevant data for each element type.
- Spring, seat-belt and stonewall transient data may be extracted from the **.XTF** file via the Oasys T/HIS time-history plotting programme.
- Velocity of moving stonewalls can be deduced from the velocities of the optional extra nodes that can be placed on such stonewalls.

12.15 Theory and Formulae

This section describes some of the theory and equations use to process data components in D3PLOT.

12.15.1 Manipulations of stress and strain tensor components

We have adopted the tensor notation for global stresses :

$$[S] = \begin{bmatrix} \sigma_x & & \\ \tau_{xy} & \sigma_y & \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix}$$ (Symmetric: Upper triangle = lower triangle)

Where σ_X, σ_Y, σ_Z are the "direct" stresses, and τ_{XY}, τ_{YZ}, τ_{XZ} the shear stresses.
Similarly we have adopted the notation for global strains:

$$[E] = \begin{bmatrix} \epsilon_x & & \\ \epsilon_{xy} & \epsilon_y & \\ \epsilon_{xz} & \epsilon_{yz} & \epsilon_z \end{bmatrix}$$ (Symmetric: Upper triangle = lower triangle.

Where ϵ_X, ϵ_Y, ϵ_Z are the "direct" strains, and ϵ_{XY}, ϵ_{YZ}, ϵ_{XZ} the shear strains.

12.15.1.1 Rotating a tensor to give element local stresses

If we have a set of direction cosines in the 3x3 matrix **[R]** then we can rotate a tensor thus:

$$[S'] = [R][S][R]^T \text{ where } [S'] = \begin{bmatrix} \sigma_x & & \\ \tau_{xy} & \sigma_y & \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix}$$

This is how the global to local transformation of stresses and strains is carried out when the **FRAME_OF_REFERENCE** is set to **LOCAL**: the **[R]** matrix is formed from the local axes of the element. The prime " ’ " notation is used to signify that the component is in the local (as opposed to global) coordinate system.

12.15.1.2 Computing von Mises Stress The deviatoric stress.

This is given by:

$$1/\sqrt{2}\ [(\sigma_X-\sigma_Y)^2 + (\sigma_Y-\sigma_Z)^2 + (\sigma_Z-\sigma_X)^2 + 6(\tau_{XY}^2 + \tau_{YZ}^2 + \tau_{XZ}^2)]^{1/2}$$

12.15.1.3 Computing **PRESSURE** The hydrostatic pressure.

This is given by: $\sigma_{-(X} + \sigma_Y + \sigma_Z) / 3.0$ (Note: compression +ve)

12.15.1.4 Computing **PRINCIPAL** stresses

The principal stresses (maximum, middle, minimum) are the three roots (**P**) of the cubic:

$$P_{MAX}, P_{MID}, P_{MIN} = P^3 - (\sigma_X + \sigma_Y + \sigma_Z)\ .\ P^2 + (\sigma_X\sigma_Y + \sigma_Y\sigma_Z + \sigma_Z\sigma_X - \tau_{XY}^2 - \tau_{YZ}^2 - \tau_{ZX}^2)\ .\ P - (\sigma_X\sigma_Y\sigma_Z + 2\tau_{XY}\tau_{YZ}\tau_{XZ} - \sigma_X\tau_{YZ}^2 - \sigma_Y\tau_{XZ}^2 - \sigma_Z\tau_{XY}^2) = 0$$

Where $P_{MAX} > P_{MID} > P_{MIN}$

Deviatoric principal stresses values are given by subtracting the hydrostatic pressure, thus:

[P_N - PRESSURE]

12.15.1.5 Computing **MAX_SHEAR_STRESS**.

This is given by: $(P_{MAX} - P_{MIN}) / 2.0$

12.15.1.6 Computing equivalent strain values.

These are all computed in exactly the same way as the stress terms above, substituting **[E]** for **[S]**, ***except*** that the von Mises strain has a factor of 2/3 applied:

This is given by:

$$2/3\sqrt{2} * [(\epsilon_X-\epsilon_Y)^2 + (\epsilon_Y-\epsilon_Z)^2 + (\epsilon_Z-\epsilon_X)^2 + 6(\epsilon_{XY}^2 + \epsilon_{YZ}^2 + \epsilon_{XZ}^2)]^{1/2}$$

13 D3PLOT USE OF GRAPHICS HARDWARE

D3PLOT supports 2 categories of graphics devices:

X_Windows	**X(option)**	2-D windows on any Unix hardware, and on PCs via emulation.
OpenGL	**OPENGL**	3-D windows on all common hardware. Only available method under Windows.

This section gives more details about these device categories, and describes what capabilities are available on each one. You should not normally need to read this section, and it is included for interest only. If you have problems with graphics hardware please contact Oasys for advice.

13.1 The "X" (X_Windows) 2-D protocol.

X_Windows is the most widely supported graphics protocol on modern engineering work-stations. It has the following attributes:

- It uses a client/server mode of working that makes it network transparent. That is you can display results on one machine (the server), while actually running on another, (the client). Client and server may be totally different machines connected by a network, or indeed the same machine working autonomously.
- It supports a wide range of screen types. Almost any graphics screen of any resolution and type will work under this protocol.
- It integrates well with window managers. All current workstation window managers will support multiple "X" windows.
- It is two dimensional only. (The PEX 3D extension has proved unsuccessful in the face of competition from OpenGL.)

The D3PLOT screen menu is an X-Window, a child of the window manager, and it has its own children that it manages locally.

The menu system always uses the default visual of the screen so that, in itself, it will not clash with other applications. The graphics sub-window within this menu may also be invoked with the default visual, or with a different one.

X "visuals" are discussed in the next section.

13.1.1 X_Windows colour visuals and their attributes

Since the X_Windows protocol is designed to run on a wide range of hardware it offers a range of four colour "visuals" to employ the various graphics screens to best effect.

VISUAL TYPE	Typical #Bit-planes	#Colours available	Colourmap type
PseudoColor	4 - 12	16 - 4096	Read/write
StaticColor	4 - 12	16 - 4096	Read-only
DirectColor	4 - 24	16 - 16777216	Read/write
TrueColor	4 - 24	16 - 16777216	Read-only

You don’t have to understand X visuals fully to use D3PLOT, but you should be aware that the visual type and number of bit-planes you use have an influence on image quality, interaction between the various windows on the screen and animation speed.

#Bit-planes: The number of colours available is $2^{\#BIT\text{-}PLANES}$. Most display modes in D3PLOT will function with 16 colours, ie 4 bit-planes; but the two lighting modes **SH (GREYSCALE)** and **SI SHADED_IMAGE** need at least 100 colours to give decent results, and work best with 256 or more colours. So using more bit-planes will give better quality images: on a 24 bit-plane visual D3PLOT will give "true" colour rendering.

However animation requires images in memory (pixmaps) to be transferred to the screen, and the greater the number of bit-planes in an image the longer this takes. So using more bit-planes gives slower animation.

Map type: Read/write colourmap visuals (Pseudocolor and Directcolor) permit colours to be changed dynamically on the screen without re-drawing, whereas read-only ones require the image to be re-drawn if colours are to be changed. In addition read/write colourmaps permit entries to be created that match as exactly as possible the shades required, whereas read-only maps have to select shades from those available: this is not a problem on 24 bit-plane screens where all possible shades exist, but it can give inferior lighting plots on devices with fewer bit-planes.

More significantly read/write visuals generally require multiple screen windows to share colourmaps, and this can lead to colours in some windows changing as entries which are correct for one window conflict with those for another. Read-only maps never suffer from this problem.

13.1.2 Choosing an X_Windows visual

From the above discussion it should be clear that there is no "best" X_Windows visual for D3PLOT, and indeed a high performance 24 bit-plane visual may be a liability if fast animation is required. Therefore D3PLOT release 8.0 allows you to choose any visual supported by your server. When it asks you to give a device type:

You may select one of the following X options:

X8 Using an 8 bit-plane visual for fast animation.

D3PLOT will use a TrueColor visual if it exists, failing that then PseudoColor and finally StaticColor. Shading and lighting should be acceptable, but dithering will be required to reproduce all shades.

X24 Using a 24 bit-plane visual for best quality lighting and shading.

D3PLOT will choose a TrueColor or DirectColor visual. Shaded and SI plots will be of good quality, but animation will be slower as three times as much information has to be moved around.

XMENU Interactive selection of visuals from menu.

This option presents you with a menu of the visuals available on the server and lets you select one. Considerable on-line help is available.

X8 will be adequate for most users, **X24** is only justified if you are going to be grabbing screen images and need the higher image quality.

Note that "dithered" laser plots are sent to the screen at the depth of the visual, but are always generated in "true" (effectively 24 bit-plane) colour in the laser file.

13.2 3D protocol: OpenGL.

The OpenGL graphics library has become the de-facto industry standard, and is available on virtually all hardware.

OpenGL itself has no native window support, it runs under the windowing system of its host machine. Therefore on Unix machines OpenGL rendering takes place within an X window, and on PCs within a "windows" window. In environments that support it OpenGL can operate in separate client/server mode, and using "objects" in the server can be efficient in this mode.

OpenGL is similar in most respects to X_Windows except that it is fully three-dimensional and utilises hardware acceleration for most graphics functions. In particular:

- The image can be rotated, translated and scaled dynamically using the mouse as under X. But this is now carried out by the hardware so, depending on hardware power and size of model, this can be done far faster.
- All calculations involving hidden-surface removal are done by the hardware using Z-buffering, generally far faster than they can be achieved in software.
- Likewise all shading and lighting calculations are done in hardware, again far faster than in software. Smooth (gouraud) shading and transparency are implemented in hardware.
- "Clipping", the ability to calculate the intersection between the image and arbitrary planes, is provided. This provides facilities such as "Z-clipping" which are not available in software.

OpenGL will give better performance than 2D X under nearly all circumstances. The exceptions to this are:

Large animations:	3D animations require up to4 times the amount of memory as the equivalent under X. If you attempt to create large animations you may run your machine out of memory, or at least cause it to "page" unacceptably. Very large (ie product of #vectors x #frames) animations are best carried out under X for this reason.
Hidden-line plots:	Hidden-line (and line contour) plots produced by Z-buffering are not as good as those generated in software. You can get round this by switching temporarily back to 2-D mode.

The complications of different visuals and numbers of bit-planes do not usually apply to OpenGL. They operate in "RGB" mode (roughly equivalent to Truecolor), and generate intermediate shades by hardware dithering if not enough bit-planes are available to produce the required colour directly.

If your device supports 3D you should use it.

13.3 Summary of capabilities of each graphics protocol

The following table summarises the capabilities of the various graphics devices listed in this section.

FUNCTION	**X-Windows**	**OpenGL**
Display mode:		
LI (**LINE**)	x	x
HI (**HIDDEN_LINE**)	x	x (1)
LC (**LINE_CONTOUR**)	x	x (1)
CT (**CONTINUOUS_TONE**)	x	x
VE (**VECTOR**)	x	x
ARROW	x	x
SH (**GREYSCALE**)	x (2)	x
SI (**SHADED_IMAGE**)	x (2)	x
Function:		
Animations	x	x (3)
Plots may be stopped	x	x
Dynamic viewing	x	x

Notes:

(1) Hidden-line and line contour quality under 3D is adequate. Better results may be obtained by switching temporarily to 2D mode.

(2) Quality on visuals with fewer than 8 bit-planes not good.

(3) Very large animations under 3D may run the computer out of memory. Switching to 2D mode may be better for these.

Some minor differences between output format on different devices may be found, for example character sizes, but generally plots should look the same on all devices.

14 PROBLEM SOLVING

This section describes some common problems, and gives suggestions about solving them. It doesn't attempt to cover all possible causes: please call Oasys if you cannot solve your problems.

14.1 Problems reading files:

The files are read but seem to be corrupt on initialisation.

If the node and element numbering is reported as being scrambled it is likely that the analysis job crashed on initialisation or, if the job is still running, it may not yet have initialised. Check to see if it crashed (see the **.LOG** file). If it is still running try reading it again later when it has got a bit further (this is because the node and element arbitrary numbering tables are the last items to be written at job initialisation).

Alternatively the automatic file format detection may have mis-diagnosed the file's data format. Check that the **FILE_TYPE_32** or **_64** environment variables are set correctly (Section 12.1.4). This is particularly the case when reading older 64 bit files.

D3PLOT crashes when reading in a file.

This generally means that the **.PTF** file is corrupt. For a job that is still running wait for it to be initialised, for a job that has finished this means that it crashed during initialisation leaving an incomplete file. The D3PLOT crash is due to trying to work out coordinate limits from nonsensical values.

The crash can also be caused by trying to read 64 bit files in **IEEE** rather than **Cray** format, or vice-versa. See Section 12.1.4.

The last state(s) in the file seem to have nonsensical times and/or corrupt results.

This usually means that a complete state has been only partially written. If the analysis job is still running it usually means that the computer system buffers are still holding data waiting to be written to disk. Wait to see what happens when the next state is written as this will probably sort out its predecessor. You can force a dump of a plot state by using the **STATUS** command in the **OASYS_81** command shell.

If the job has finished then it probably crashed or ran out of disk space. Check the **messag** or **.otf** files for error messages, check the **.log** file for evidence of crashes or disk space exhaustion.

D3PLOT reports that the **.XTF** or **.CTF** files are incompatible with the **.PTF** file.

This means that the control parameters in the various files suggest that they have come from different analyses. The most common cause of this is that files from an old analysis of the same name have not been deleted when the new one is run: check the creation dates on the files.

D3PLOT will ignore the incompatible **.ctf** / **.xtf** file(s) and continue running.

D3PLOT issues warning that >1000 element meet at a node

This invariably means that the file is either corrupt (ie completed analysis job crashed on initialisation), or is still waiting to initialise (job still running). Take action as described for when D3PLOT crashes reading a file.

D3PLOT issues warning that solid or thick shell elements have crossed faces.

This may not be an error: it is possible (but unusual) for this situation to arise legitimately. If this is the only message issued and D3PLOT initialises normally have a look at the offending elements and check their topology. If this is blatantly wrong it means that the file is corrupt.

D3PLOT warns that 6-noded thick shells have mis-numbered faces.

It is a common error to create triangular thick shells using the solid "wedge" numbering sequence (1,2,3,4,5,5,6,6) instead of the "extruded triangle" sequence (1,2,3,3,4,5,6,6). LS-DYNA will still run, but the results for these elements will be dodgy. D3PLOT will also run, but the face numbering for these elements will be wrong leading to mis-diagnosis of external faces, free edges and data averaging at nodes.

D3PLOT issues warning that duplicate solids or thick shells exist

Again this may indeed be the case. Check as described above, and take heed that coincident solids may not be displayed unless the **DISPLAY_OPTIONS**, **INTERNAL_FACES** switch is turned **on**. (Since all their faces will be marked internal.)

Problems with missing **.PTF** file family members: last <n> states not read

If the final <n> states from your analysis are not read, and these appear to be in the last family member(s), check for gaps in family member numbering. If one or more family members appear to be missing check the reason. However you can skip gaps in the family member sequence using the **FILE >**, **FILE_SKIP** command: see Section 4.1.7.

D3PLOT fails to read some states from a file, with a regular pattern

This can occur if the family member size has been set to a value smaller than that used when writing the files, as states in the tail end of family members are not read.

In D3PLOT 8.0 onwards family member size detection should be automatic, but you can over-ride it using the **FILE >**, **FAM_SIZE** command, or by setting the **FAM_SIZE** environment variable as described in Section 4.1.7.

File protection problems.

D3PLOT does not require "write" access to database files, so you can process results to which you only have "read-only" access.

However operations that create files: laser plotting, view storage and session file generation, all require "write" access to the relevant directories. If a file open fails due to protection errors you will be warned and the operation will be aborted.

14.2 General graphics problems:

The terminal won’t draw anything

There are several possible causes. Have you:

- **BLANKED** everything out of sight? Try turning the **BLANKING** switch off.
- **CLIPPED** everything out of sight? Try turning the **VOLUME_CLIPPING** switch off.
- Turned all entities off? Check the status of the **ENTITY** settings.
- Scaled, rotated, zoomed everything out of sight? Try a **ZERO** command.
- Are your deformations so large that the image at this complete state time is off the screen? Try an **AC** command.
- Got a corrupt **.PTF** file? (Job crashed)

Facets of solids and thick-shells disappear when the model is very distorted.

This is because the "back face" detection algorithm can get confused when elements become very misshapen, and removes faces that it shouldn’t. Try turning the **DISPLAY_OPTIONS**, **BACK_FACE** switch on, which should cure the problem. Alternatively you can reduce distortions artificially by setting the displacement magnification factors to values less than 1.0: see **DEFORM**, **MAGNIFY_DISPLACEMENTS**.

2-D hidden-line removal seems to make mistakes.

The default "painter" algorithm is cheap and cheerful, and it can make mistakes. You can prevent this by using the more expensive "rigorous" algorithm: see **DISPLAY_OPTIONS**, **HIDDEN_LINE_OPTIONS**.

Problems displaying stonewalls

Stonewalls, especially infinite ones, present a problem since their characteristic size is so much greater than typical facets in a model. This can show up very clearly in hidden-line plots where they often appear to be in front of or behind where they should be.

This is usually due to their interaction with the perspective calculation, and turning this off (**PERSPECTIVE**, **OFF**), and switching to "rigorous" hidden-line removal will improve matters considerably.

14.3 Memory consumption problems.

The message “unable to obtain more memory” is given

This means that an attempt to allocate more memory has failed because the system has refused to give it. This can happen in many different contexts, but is most common when building animations under OpenGL, as this is a memory-intensive process.

The problem may be soluble at the system level:

(1) All systems

- Check that other unnecessary processes competing for system resources have been shut down. Use the **`ps`** command (Unix) or the Task Manager (Windows NT) to examine system usage.
- Check that you have enough swap space configured. (System administrator privileges are required to alter this.)

(2) Unix systems: Make sure that the operating system is not imposing arbitrary limits.

- Artificial limits may be imposed in your command shell. The **`unlimit`** command can be used to lift all restrictions that you, as a user, have privileges to change.
- The maximum "data segment size" in the kernel may be set to a low value. Many Unix systems come configured with this set to their physical memory size, which stops a given process spilling far into swap space. You will need system administrator privileges to change this as it requires the kernel to be modified and rebuilt.
- Check that the window manager process has not been running for a long time, and has accumulated a lot of memory. If it has it may be necessary to kill it (ie revert to console mode) then restart it using **`startx`**.

If these do not work then you will have to reduce the amount of memory you are using within the D3PLOT process itself:

- Use the **FILE >MEMORY** then **VIEW** database command to empty (partially or fully) the database.
- Change the animation mode from **VECTOR** to **DIRECT** (**ANIM >**, **DISPLAY_MODE**: see Section 4.4.2.2) to save memory.
- If you have two computers try using client / remote server mode. See Section 4.4.2.2.
- See Section 11.7 on "memory management"

14.4 Graphics problems peculiar to X_Windows devices.

The programme seems to “hang” when first started

Possibly there is a communications problem, or you have given the wrong display address for the **`DISPLAY`** environment variable (in which case your window may be appearing on another machine!) Failing that try restarting the programme as window managers can occasionally "hang".

The message “**`Installed private map`**” appears when the window is created.

This is not an error, but it may be the prelude to problems. What it indicates is that D3PLOT is unable to find enough free colour cells in the current, shared colour map and has decided to create its own. This may cause colours to change in other windows. If this is the case you could try requesting fewer colours or running under a different visual: see Section 10.1.

Colours in other windows change as I run D3PLOT.

This is a symptom of the private colourmap described above: it occurs when a Pseudo- or DirectColor visual is run with a private (non-shared) colourmap. This ’colourmap flashing’ can occur on older displays which do not have

adequate hardware support for multiple windows using different colourmaps. The problem can sometimes be solved by:

- Setting the environment variable **SM_USE_VISUAL** to **default** (C shell syntax)
  setenv SM_USE_VISUAL default (Bourne/Korn shell syntax)
  SM_USE_VISUAL=default; export SM_USE_VISUAL
- Selecting device **XMENU...**, then choosing the visual that is marked as "**default**".

If these remedies do not work please contact Oasys for further advice.

Colours in D3PLOT graphics window correct only when I “zoom” or screen-pick

This happens when a private colourmap is installed as described above on low-end devices that cannot support mulitple maps. The best solution is to avoid the creation of a private colourmap using the strategies referred to above, but if this unavoidable then it may help to change the default settings of the window manager.

In your home directory there needs to be a file "**.Xdefaults**". (Use **ls -a** to see "hidden" files starting with a "**.**") Edit this file to contain the line:

```
*colormapFocusPolicy: pointer
```

Note that the case of letters in both filename and edited entry is important and must be exactly as shown here. When you have completed this log out completely from the screen and then log back in again to make it active. (It is not sufficient just to restart the window manager.)

When an animation is reloaded the colours are all wrong

In versions of D3PLOT prior to version 6.0 the colourmap was not saved in animation files. Release 6.0 onwards remedies this problem.

The message “**Found unknown code nnn in stack**” appears while animating.

Animations are stored in a memory "stack" as a series of codes that represent the graphics operations required to build the image. Occasionally a corrupt code finds its way into this stack and causes this message.

The most common cause is when the X server has become overloaded and lost synchronisation with its client process. You may need to slow down animation to prevent this..

The image doesn’t change scale when the window is resized.

Occasionally the resize "event" is missed by the D3PLOT command handler. If this happens just resize the window again by a tiny amount, to create a new "event", which should then be processed.

Problems with X_Windows addresses over a network

The X_Windows protocol makes a distinction between the process running the computer programme, the "client", and that displaying the graphics, the "server". This is to enable a software package to operate in the same way whether it is running on the local machine, or on a remote machine over a network.

Where client and server are the same machine then no confusion arises since there is no ambiguity about where graphics should be displayed. However when you are displaying results over a network from a remote client you may get the message:

```
Unable to open display :0
```

This means that the host machine must be told where to display graphics. On UNIX machines you do this by setting the **DISPLAY** environment variable as follows:

C shell: `setenv DISPLAY <server>:0`

Bourne shell: `DISPLAY=<server>:0 export DISPLAY`

Where **`<server>`** is the network address of the machine on which you are displaying the graphics. A raw address (eg **`69.60.10.1`**) may always be used or, if the host machine knows about your server, you can use the machine's name.

Multiple screens: The "**`:0`**" is the screen id on that machine, and this will almost always be screen zero as here. The exception is when you have multiple screens attached to a device, in which case the syntax will become "**`:0.0`**" for the first screen, "**`:0.1`**" for the second screen, and so on.

Therefore typical destination commands might be:

`setenv DISPLAY snoopy:0`	(Machine "snoopy", screen 0)
`setenv DISPLAY 69.177.15.2:0`	(Address 69.177.15.2, screen 0)
`setenv DISPLAY :0.1`	(This machine, screen 1)

(**nb:** On UNIX machines see the file **`/etc/hosts`** for host names and internet addresses. You will only be able to refer by name to machines in this file. However you can display on any machine by referring directly to its numeric address: the names in this file are just convenient aliases for this.)

Problems when the server refuses a connection

A further complication may arise due to permission not being granted for that host to connect to your server. This will generate the message:

`Host is not permitted to connect to server`

In this case you must also tell your server to accept graphics from the host with the command:

`xhost + <hostname>`

If you omit **`<hostname>`** then access will be permitted to all hosts. Your system manager can configure your system such that access is permanently permitted to a selected list of hosts, or indeed all hosts. The latter (all hosts) is potential security hole since the inter-client communication permitted by X window managers is a possible back door for hackers. Network users: you have been warned!

Next section

14.5 Graphics problems peculiar to OpenGL

Screen-picking selects invisible entities

Because hidden surface removal is performed in the hardware Z-buffer it is difficult for the client D3PLOT process to know what is and is not visible in a hidden or shaded plot. The picking algorithms attempt to work this out, but occasionally they make mistakes.

There is no absolute cure for this, but suggestions are: reject the pick, rotate the image a touch, and try again as this may make the selection of the visible element less ambiguous. Or maybe just try again taking more care to pick the centre of the item you want. Failing that swap back to 2-D mode (**M2D**), and generate the image in software.

The image disappears altogether

This can happen for a variety of reasons:

Z-clipping	It is possible to rotate, translate or scale an image such that it passes totally outside the screen Z clipping region. Try resetting these planes: (**3D_OPTS**, **Reset Z**). Turn the projection box (**3D_OPTS**, **SHOW_PROJ** switch) on to see the view frustrum and clipping plane locations.
Bad matrix	The hardware rotation matrix can sometimes get corrupted if a very large number of screen rotations give rise to progressive ill-conditioning: (ie direction cosines no longer have unit length). In this case reset the matrix with the command **ZERO** (reset to default viewing state).

There can also be less obscure reasons: see "The terminal won’t draw anything" in Section 13.2.

Hidden-line quality is poor in 3-D mode

This is a limitation of Z-buffered graphics. The problem arises because hidden-line plots are generated by drawing visible borders around black polygons, and sometimes the Z coordinates of the polygon win, and sometimes those of the line win, resulting in a broken line. The breaks will move as the image is rotated. D3PLOT attempts to get round this by raising the lines slightly above the polygon surface, but this doesn’t work very well when polygons are near edge-on to the viewer.

You can adjust the amount by which the polygon borders are lifted above the surface by holding down the <**left control**> button and the <**left mouse button**>. Moving the mouse up the screen will lift lines off the surface towards you, moving it down will move them back down.

The best solution is to switch back to 2-D mode temporarily as the software algorithms can avoid this problem.

Animations in 3D mode become horrendously slow.

Storing animations in 3D mode can take up to 4 times as much memory as in 2D (or X) mode: see Sections 4.4.2.2 and 4.4.3. This means that you run your machine out of memory much faster and the slowdown is due to it "paging" memory to and from disk.

Unless you are prepared to buy more memory the only thing you can do is to reduce the memory being used by one or more of:

- Reducing #frames and/or #vectors by simplifying the plot.
- Using a "cheaper" display mode.
- Switching to **DIRECT** animation mode - slower, but uses no memory.
- Emptying some of your database memory (**FILE >**, **DATABASE**, **EMPTY_xxx**)

Alternatively you could use 2D mode (**View Box**, **M2D** command). Animations in 2D use a lot less memory although dynamic viewing and changes to the image become much slower to implement.

14.6 Miscellaneous problems.

Problems with coincident solid or thick-shell elements.

You are warned if coincident solid or thick-shell elements are found when the files are read in. This is because all faces of such elements will be flagged as "internal" and the elements won't be displayed unless you turn the **DISPLAY_OPTIONS**, **INTERNAL_FACES** switch **ON**.

There may also be problems with averaging data at nodes for such elements unless you set the **AVERAGING: MATERIAL_IGNORED** and **BLANKING_IGNORED** switches carefully. See Section 4.2.4.5 for more details.

You can list coincident elements of all types with the **WRITE**, **COINCIDENT** command.

Problems displaying "extra nodes on rigid bodies"

D3PLOT considers nodes that are not attached to elements to be non-structural and does not normally display them. "Extra nodes on rigid bodies" can fall into this category as, unless they are attached to an element, they do not appear to be part of anything. To see these you need to turn the **ENTITY**, **ALL_NODES** switch **ON**.

You can list nodes that D3PLOT considers to be non-structural with the **WRITE**, **UNATTACHED_NODES** command.

Problems seeing contact surfaces.

You can only see your contact surfaces if you have written the (optional) contact force (**.CTF**) file during your analysis. No **.CTF** file: no contact surfaces visible.

If your analysis would take a long time to re-run you can at least visualise the contact surface geometry by reinitialising it under a different name with a contact force file, and then renaming this **.CTF** to look as if it came from the original analysis. D3PLOT will extract the geometry from this but obviously will be unable to read transient contact force data which isn't there. It can cope with this, although you will get the warning message at initialisation that there are no contact force states to match structural states.

Problems seeing springs, joints and stonewalls.

The topology of these elements is extracted from the **.XTF** file so, if it isn't there, you won't see these.

Reinitialising and renaming the **.XTF** file as above will solve this problem.

14.7 **MEMORY** Viewing and controlling the memory usage for this process, and the whole machine.

On all operating systems D3PLOT has to co-exist with other processes, and when its consumption of system resources is small this is not an issue. However as you process larger models you will approach the capacity of you machine, and control of memory use will become important. There are two key issues:

Process size, the size of the D3PLOT process, affects performance.

- As this approaches 80 - 90% of the physical memory size of the machine it will become necessary for the operating system to use "virtual" memory. This is because other processes need to maintain a presence in memory and, to make space, some of the D3PLOT process will need to be paged ("swapped out") onto virtual memory on disk.
- Paging, when some of the process pages[(1)] are swapped onto disk, has a major impact on speed because of the delay in reading them back into memory again when needed. The symptoms are a much slower response, accompanied by the disk chattering away. However the process will continue to run, albeit more slowly.

Virtual memory usage - running out will halt processes, or even crash the machine.

- Operating systems maintain "swap space", which is an area of disk set aside for pages of memory that have been "paged" out of physical memory. This is known as "virtual" memory, and is usually up to 3 times the size of physical memory, but can be any value set by the system administrator.
- Virtual memory allows computers to handle the situation where the sum total of memory space requested by all processes is greater than the physical memory available. They do this by swapping unneeded pages of memory onto disk, then swapping them back into memory (evicting other pages to make space) when they are required again. A "page fault" occurs when an executing process requires a page that is not resident in memory and has to be read in from disk.
- Parking dormant processes on disk has no impact on the performance of running ones. But it will be obvious that if a running process makes excessive use of virtual memory it will generate a lot of page faults, which will slow down its performance.
- If virtual memory space is all used up then the operating system will start to kill processes, and the machine may crash if essential services cannot obtain the memory they need!

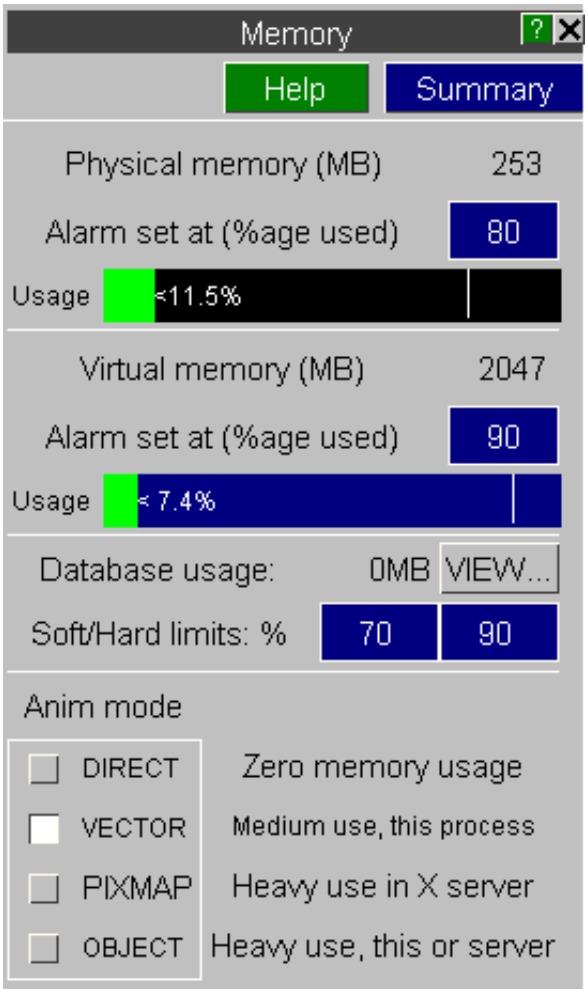
The **MEMORY** panel itself is shown here.

This section shows Physical memory statistics.
The machine has 253MB of memory. D3PLOT is using 11.5%. The alarm has been set at 80%.

This section shows Virtual memory statistic s.
The machine has 2047MB of swap space, of which 7.4% is in use, alarm set at 90%.

This section shows the results database memory usage summary.

Finally this section allows you to alter the graphics display mode. (Some modes use more memory than others.)



Memory usage bars:

The two usage bars (%age physical memory used by D3PLOT , and **total** %age virtual memory usage over the whole machine) are replicated in miniature on the **MEM** button on the front panel for easy reference. So long as they both stay in the green you don't have a problem, and even light orange (60%) is probably OK. More than that and you may need to take action - see below.

Alarm limits:

Both physical and virtual limits have alarm values set at 80 and 90% respectively. If an alarm limit is reached D3PLOT stops what it is doing and maps an alarm panel explaining the problem. You can change alarm values at any time, and set them to large values (eg 1000%) if you want to ignore them altogether.

Database Usage:

This summarises the settings of the results database, and the **VIEW...** button takes you to the main database manager.

Allows you to select an animation mode as described in Section 4.4.2.2. (The same as **ANIM > DISPLAY_MODE**). Some modes use more memory than others.

What to do if memory runs short.

- If virtual memory is short see if there is anything else running on the machine that you can shut down (remember the virtual memory bar is for the whole machine, not just D3PLOT).
- This usually happens when building animations. See if you can cut down what is displayed to save memory:
- Display alternate states instead of every state;
- Use a simpler display mode (eg flat shading, not smooth; **SI** not **CT**)
- Turn off unneeded display items (node symbols, labels)
- If the results database is very full use its **OPTIONS > EMPTY_xxx** commands to release some space, and reduce the soft and hard %ages. (This may not release space back to the operating system, but the space freed can be re-used internally.)
- Use a less expensive display mode:
- **DIRECT** uses no display memory, but is slow.
- **VECTOR** is a reasonable compromise (the default)
- **PIXMAP/OBJECT** use a lot of display memory in the server process

In extreme cases you may find that none of the above free enough memory and you have to exit and start again with "cheaper" (in terms of memory) settings. If this happens:

- Before reading in a model use the **MEMORY** button on the filename panel to select the display mode and database soft/hard %ages.
- Then define a filename and read in the model. The new settings will be used ab initio and will result in lower memory usage.

Another problem that can occur following **PIXMAP** or **OBJECT** mode animations is that the graphics server process (owned by the operating system) may grow to a very large size and cause problems. This is because some operating systems don't allow memory to be released once it has been allocated.

It will be necessary to shut down and restart this process in the following way:

- Logout from the display.
- At the "login" prompt choose the "no windows" or "console" mode (usually under an "options" menu). This will shut down the X server completely.
- Most "Common Desktop Environment" (CDE) window managers will automatically restart the X server after a short delay.

Dealing with huge models by using two computers in client / remote server mode.

If, despite everything you do, your model is too big to fit into your machine you may still be able to deal with it by splitting the job over two machines. The "client" machine runs the D3PLOT process, and the remote "server" displays the graphics.

- On the client set the **DISPLAY** environment variable to point to the display on the server machine. (Eg **setenv DISPLAY remote_name:0**)
- Under X-windows use **PIXMAP** mode, and under OpenGL use **OBJECT** mode. Both of these result in the graphics data being displayed in the server, not the client.
- The initial transfer of graphics data from client to server will be slow, since it has to go down a network. But subsequent animation, performed locally on the server, will be fast.
- Keep and eye on the memory usage on the server: D3PLOT does not monitor this for you. (It only "knows" about the client machine.)

Section 4.4.2.2 explains how animation works in client/server mode. Sections 2.3 and 2.4 give more information on specifying network addresses.

1. A "page" of memory is a conveniently sized quantum, typically 4kB: this varies between systems.

Appendices

APPENDIX I PROGRAMME LIMITATIONS

D3PLOT has certain inherent limitations because of its internal structure. These are:

Maximum number of nodes, elements and materials:

Maximum number of nodes is	$2^{**31} - 1 = 2.1e9$	
Maximum number of elements is	$2^{**31 - 1 = 2.1e9}$	(Under 2D X11 graphics $2^{**24} - 1 = 16.7e6$)
Maximum number of materials is	$2^{**31 - 1 = 2.1e9}$	(Under 2D X11 graphics $2**^{16} - 1 = 65,535$)

These are the maximum ***quantities*** that may be processed. The maximum external label ids are limited only by output field width, so that the permitted external label range is 1 - 2**31 (approx 2e9, the same as LS-DYNA).

Maximum problem size:

Memory management in D3PLOT is "dynamic". This means that space is requested from the operating system as it is required and, theoretically, problems up to size limits above can be processed. The only realistic limit will be the ability of your computer to provide memory and processing power.

In practice a typical small workstation will easily process models of 500,000 elements, and addition of more memory and page file (swap) space will raise this limit to several million elements. For example processing a model with 170,000 shells on the author’s machine required a total of about 55MBytes of memory. However large animations consume memory, and this may impose a limit upon effective problem size.

Most 32 bit operating systems impose a limit of 2GBytes on any process size, and this will tend to impose the upperbound on these systems. This is due to the maximum memory addresses can be stored in a 32 bit integer (4GB), and the needs of the operating system itself. An exception is x86 hardware under Windows and Linux which, with suitable configuration, will allow 3GB of memory to be used.

On 64 bit systems maximum memory usage is effectively unlimited, since a 64 bit integers permit up to 1.8e19 bytes (16,000,000 Terabytes) to be addressed. The maximum process size will be dictated by the amount of physical and virtual memory available.

Maximum animation sequence.

Up to 2^{**31} (roughly 2e9) frames may be generated, saved and displayed - subject to memory limitations.

The amount of information, the product of <amount of graphics information> x <number of frames>, is limited only by the amount of memory available.

APPENDIX II OA_PREF FILE: SETTING USER PREFERENCES

It is possible to set "user preferences" in the **oa_pref** file outside the programme. This file allows customisation of many aspects of the programme: for example colours, function key operations, display attributes, laser file settings, and so on. This can be done in three ways:

1. From Options > Edit Prefs in D3PLOT
2. From the Preferences editor called from the Oasys shell
3. By hand-editing the preference file.

Where the **oa_pref** file is located, and search order.

The "oa_pref" file is read once when D3PLOT starts, and is searched for in the following locations in the order given:

(1) The **$OASYS** directory. Qualitatively a "site wide" set of options.

(2) Your home directory (**$HOME** on Unix/Linux, **$USERPROFILE** on Windows.). Qualitatively an individual user's set of options.

(3) The current working directory. Qualitatively a job-specific set of options.

Each file is read if found, but if a definition occurs in more than one file then the most recently encountered definition is used. Therefore you can have a local definition in your $HOME or $cwd directories that supersedes the "global" one in $OASYS, but still read the latter file.

oa_pref file naming conventions.

"oa_pref" file naming convention.	
In the **$OASYS** directory	"**oa_pref**" in all cases (The use of ".oa_pref" with a leading "." on Unix/Linux is deprecated.)
In **$HOME** directory (Unix/Linux) In **$USERPROFILE** (Windows - typically **C:\Documents and Settings\<***user id***>**)	
In current directory ($cwd)	

How to determine the value of $USERPROFILE on Windows.

Normally this will be located in **c:\Documents and Settings**<*user id*>**** however to find out this value on a Windows machine do the following:

- Bring up an MS-DOS window (*Start menu*, *Accessories*, *Command Prompt*)
- Type **SET** in this window to receive a listing of all environment variables.

(For reasons best known to Microsoft not all environment variables appear in the "Environment variables" window accessed from *My Computer*, *Properties*, *Advanced*.... The method above is better.)

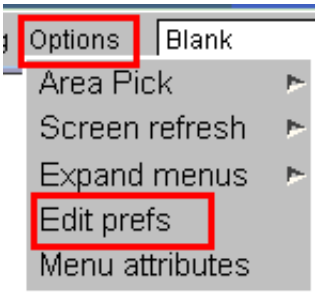
How to determine the value of $HOME on Unix / Linux

The simplest method is to give the following commands, which will jump to the $HOME directory and print its pathname:

- **cd $HOME**
- **pwd**

The interactive Preferences Editor

You are free to edit oa_pref files by hand, but there is an interactive "Preferences Editor" that may be called from within D3PLOT that makes the job much easier.

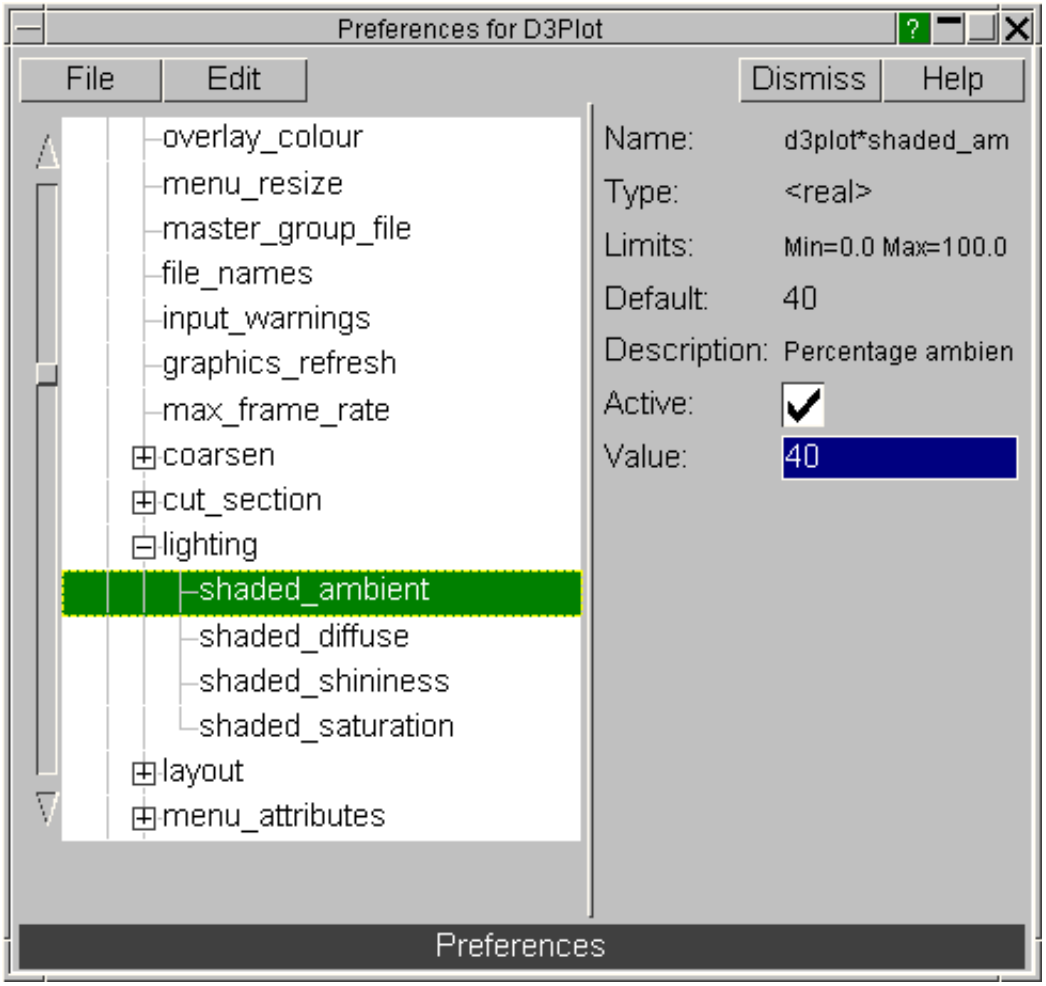


It is started by **Options, Edit Prefs:**

The preferences editor reads an XML file that contains all possible preferences and their valid options, and allows you to change them at will. In this example the user is changing the background colour in D3PLOT.

Note that changes made in the Preferences editor will not affect the current session of D3PLOT, they will only take effect the next time it is run.

If you have write permission on the oa_pref file in the $OASYS directory you will be asked if you want to update that file, otherwise you will only be given the option of updating your own file in your $HOME / $USERPROFILE directory.



In this example the user is changing the ambient light intensity.

The option is "active" (ie present in the oa_pref file) and currently is set to 40.

Usage is:
- Select an option in the Tree on the left hand side
- Make it active / inactive
- If active select a value from the popup, or type in a value if necessary

The colour of the highlighting in the left hand side tree is significant:

Green	Means that the option has been read from your $HOME file.
Red	Means that the option has been read from the $OASYS file.

In either event, regardless of the data source, the updated option will be written to the file chosen when you started the preferences editor.

Because of the order of file reading (see above), and option read from the master $OASYS file, amended, and written to your local $HOME file will take precedence when you next run D3PLOT.

Format of the **oa_pref** file

Entries are formatted in the following way: **<programme>*<option>: <setting>**

For example: **d3plot*laser_paper_size: A4**

The rules for formatting are:
- The **<programme>*<option>:** string must start at column 1;
- This string must be in lower case, and must not have any spaces in it.
- The **<setting>** must be separated from the string by at least one space.
- Lines starting with a "**#**" are treated as comments and are ignored.

(Users accustomed to setting the attributes of their window manager with the **.Xdefaults** file will recognise this format and syntax.)

Permissible "oa_pref" options for D3PLOT

The list of D3PLOT options available in this file is as follows:

Preference	Type	Description	Valid arguments	Default
contour_levels	<integer>	Number of contour levels	2 - 13	6
default_component	<string>	Valid data component name		<none>
graphics_refresh	<string>	Refresh graphics window when exposed	OFF, ON	ON
graphics_type	<string>	Graphics format to start D3PLOT with	X8, X24, X, Opengl, Default	<none>
initial_plot_mode	<string>	Initial drawing mode	LINE, HIDDEN, SHADED	SHADED
max_frame_rate	<string>	Maximum animation rate in frames/second	UNLIMITED, 1, 5, 10, 20, 25, 50, 75, 100	UNLIMITED
maximise	<logical>	Maximise window when D3PLOT started	TRUE, FALSE	FALSE
menu_resize	<string>	Which panel borders are free to resize	NONE, L, R, T, B, LR, LT, LB, LRT, LRTB, RT, RB, RTB, TB, ALL	ALL
overlay_mode	<string>	Overlay drawn	OFF, FREE, ALL	FREE
placement	<string>	Location for initial window on multi-screen display	LEFT, RIGHT, BOTTOM, TOP, LEFT_BOTTOM, LEFT_TOP, RIGHT_BOTTOM, RIGHT_TOP	<none>
plot_border	<string>	Border drawn on plot	OFF, ON	OFF
show_maxmin	<string>	Max and Min data values drawn on plot	OFF, DATA, ALL	DATA
start_in	<string>	Directory to start D3PLOT in		<none>
surface_output	<string>	Surface to output results	TOP, MIDDLE, BOTTOM, MAX, MIN, ALL	MIDDLE

The following options control automatic mesh coarsening, and set a model size threshold at which this is implemented. This can be useful if you habitually process very large models and are happy to accept slightly poorer image quality in return for faster graphics. See **COARSEN** for more information.

Preference	Type	Description	Valid arguments	Default
auto_coarsen	<string>	Automatic mesh coarsening on initial read	OFF, MILD, SEVERE	OFF
coarsen_threshold	<integer>	Min number of shells for auto-coarsening	0 - 1000000000	250000

The following options define the colours used in plots, thereby affecting plot appearance.

Preference	Type	Description	Valid arguments	Default
background_colour	<string>	Background colour	WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, GREY	BLACK
overlay_colour	<string>	Overlay colour	WHITE, GREY, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ELEMENT	GREY
text_colour	<string>	Text colour	WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, GREY	WHITE

The following options permit the default cut-section method and space system to be configured. Users who wish to default to compatibility with the LS-DYNA section definition method may wish to use these options. See **CUT_SECTIONS** for more information.

Preference	Type	Description	Valid arguments	Default
cut_section_type	<string>	Default cut section definition method	OV_ORIGIN_AND_VECTORS, N3_THREE_NODES, X_CONSTANT, Y_CONSTANT, Z_CONSTANT, LS_DYNA_METHOD	X_CONSTANT
cut_section_space	<string>	Default cut section space system	BASIC, DEFORMED, SCREEN	DEFORMED
cut_section_fsys	<string>	Cut section force and moment computation system	AUTOMATIC, GLOBAL, LOCAL	AUTOMATIC

The following set of options defines how various files are handled by D3Plot. Option "master_group_file" defines a "master" group file to be read every time a new model is opened: (see GROUPS). For information regarding the various file formats corresponding to the "read_<xxx>" options, see section 4.1.

Preference	Type	Description	Valid arguments	Default
master_group_file	<string>	Valid master (ascii) groups filename		<none>
file_names	<string>	Controls input filename syntax. LSTC = d3*, OASYS/ARUP = job.ptf*	OASYS, ARUP, LSTC	OASYS
read_ctf_file	<logical>	Read CTF file	TRUE, FALSE	TRUE
read_xtf_file	<logical>	Read XTF file	TRUE, FALSE	TRUE
read_ztf_file	<logical>	Read ZTF file	TRUE, FALSE	TRUE
read_prp_file	<logical>	Read PRP (properties) file	TRUE, FALSE	TRUE
read_set_file	<logical>	Read SET (settings) file	TRUE, FALSE	TRUE
read_asc_file	<logical>	Read ASC (ascii groups) file	TRUE, FALSE	TRUE
autocreate_ztf	<logical>	Create ZTF file automatically if required	TRUE, FALSE	TRUE
primer_version	<string>	Name of PRIMER executable for ZTF file auto-creation		primer93.exe
input_warnings	<string>	Switching and location of warnings on input	NONE, DIALOGUE, MENU	DIALOGUE

The function keys F1 to F12 can have pre-defined command files assigned to them: (see UTILITIES, SETTINGS_FILE.)

Preference	Type	Description	Valid arguments	Default
f1_key	<string>	File for key F1		<none>
f2_key	<string>	File for key F2		<none>
f3_key	<string>	File for key F3		<none>
f4_key	<string>	File for key F4		<none>
f5_key	<string>	File for key F5		<none>
f6_key	<string>	File for key F6		<none>
f7_key	<string>	File for key F7		<none>
f8_key	<string>	File for key F8		<none>
f9_key	<string>	File for key F9		<none>
f10_key	<string>	File for key F10		<none>
f11_key	<string>	File for key F11		<none>
f12_key	<string>	File for key F12		<none>

The following options define the background and watermark images that can be embedded into plots. See section 7.4.

Preference	Type	Description	Valid arguments	Default
background_image	<string>	Valid background image filename		<none>
watermark_image	<string>	Valid watermark image filename		<none>

The following strings and values control laser plotting setup (see section 7.3 for more on laser plotting).

Preference	Type	Description	Valid arguments	Default
laser_paper_size	<string>	Default paper size	US, A4	A4
laser_orientation	<string>	Default page orientation	Portrait, Landscape	Landscape
laser_mode	<string>	Default laser mode	Colour, Greyscale	Greyscale
laser_insert_file	<string>	Valid filename		<none>
laser_top_margin	<real>	Top margin size in mm		10
laser_bottom_margin	<real>	Bottom margin size in mm		30
laser_left_margin	<real>	Left margin size in mm		20
laser_right_margin	<real>	Right margin size in mm		10

Window layout commands control how multiple graphics windows are positioned and sized, and give some further options. (See section 2.6.2)

Preference	Type	Description	Valid arguments	Default
window_layout	<string>	Multiple window layout type	TILE_WIDE, TILE_TALL, CASCADE, 1x1, 2x2, 3x3	TILE_WIDE
auto_hide	<logical>	Hide graphics window function bar	TRUE, FALSE	FALSE
show_tabs	<logical>	Show window tabs on panels	TRUE, FALSE	TRUE
windows_same_size	<logical>	Windows initially the same size	TRUE, FALSE	FALSE

The following **shaded_*<xxx>*** options affect lighting (**SH**aded and **SI**) plots only.

Preference	Type	Description	Valid arguments	Default
shaded_ambient	<real>	Percentage ambient light (0-100)	0.0 - 100.0	40
shaded_diffuse	<real>	Percentage diffuse brightness (0-100)	0.0 - 100.0	90
shaded_shininess	<real>	Percentage specular brightness (0-100)	0.0 - 100.0	70
shaded_saturation	<real>	Percentage colour saturation (0-100)	0.0 - 100.0	50

The following options affect the operation and appearance of linked T/HIS: (see section 6.12)

Preference	Type	Description	Valid arguments	Default
this_window_location	<string>	Linked T/HIS window location on screen	SIBLING, CHILD, DOCKED	SIBLING
this_link_version	<string>	Name of 32 bit executable for T/HIS link		this93.exe
this_link_version_64	<string>	Name of 64 bit executable for T/HIS link		this93_64.exe
this_link_timeout	<integer>	Timeout period (seconds) for T/HIS link	1 - 1000	5

The following options affect the appearance and behaviour of the graphical user interface (see section 3.7.1), left handed support, and the mouse

Preference	Type	Description	Valid arguments	Default
display_factor	<real>	Factor on display size (0.5-2.0)	0.5 - 2.0	1.0
display_brightness	<real>	Menu brightness (0.0-1.0)	0.0 - 1.0	1.0
display_saturation	<real>	Menu colour saturation (0.0-1.0)	0.0 - 1.0	1.0
font_size	<string>	Menu font size	SMALL, DEFAULT, LARGE	DEFAULT
font_type	<string>	Menu font typeface and strength	HELVETICA, HELVETICA-BOLD, TIMES, TIMES-BOLD, COURIER, COURIER-BOLD	HELVETICA
left_handed	<string>	Left handed switching of mouse and/or keyboard	NONE, MOUSE, KEYBOARD, ALL	NONE
zoom_factor	<real>	Zoom Factor for mouse wheel(0.01-0.2)	0.01 - 0.2	0.05

If a selection menu is not wide enough to display all the contents, it can be expanded automatically by the following (see section 3.7.3)

Preference	Type	Description	Valid arguments	Default
menu_expand	<string>	Automatic menu expansion on/off switch	OFF, ON	ON
menu_expand_delay	<real>	Factor on delay time before expansion	0.1 - 5.0	1.0
menu_expand_speed	<real>	Factor on menu expansion speed	0.1 - 5.0	1.0

The following options define how Javascripts are processed by D3Plot. See section 11 for further details.

Preference	Type	Description	Valid arguments	Default
script_directory	<string>	Directory in which D3PLOT looks for scripts		$OASYS/d3plot_library/scripts

The following options allow the user to change the symbols (and their quality) representing various entities which are drawn in plots. See DISPLAY_OPTIONS for further details.

Preference	Type	Description	Valid arguments	Default
abp_symbol	<string>	Symbol for Airbag Particles	POINT, CUBE, SPHERE	POINT
abp_quality	<integer>	Quality of Airbag Particle sphere symbol	1 - 5	1
sph_symbol	<string>	Symbol for SPH elements	POINT, CUBE, SPHERE	SPHERE
sph_quality	<integer>	Quality of SPH sphere symbol	1 - 5	2

The following **<*xxx*>_visibility** flags set the relevant **ENTITY** switches, and may subsequently be turned on/off manually in the normal way. The setting given here becomes the default for <reset> operations.

Preference	Type	Description	Valid arguments	Default
mass_visibility	<string>	Lumped mass visibility	OFF, ON	OFF
spring_visibility	<string>	Spring/damper visibility	OFF, ON	ON
sbelt_visibility	<string>	Seatbelts etc. visibility	OFF, ON	ON
joint_visibility	<string>	Joint visibility	OFF, ON	ON
stonewall_visibility	<string>	Rigidwall visibility	OFF, ON	ON
segment_visibility	<string>	Contact segment visibility	OFF, ON	OFF
segment_hatching	<string>	Contact segment hatching	OFF, ON	ON

Thus a typical range of settings might be:

```
d3plot*plot_border:        ON
d3plot*menu_resize:        TBR
d3plot*initial_plot_mode:  LINE
d3plot*laser_paper_size:   A4
d3plot*laser_orientation:  Portrait
d3plot*laser_mode:
                           Greyscale
```

The only one which requires explanation is

`laser_insert_file`

This is a file, in valid PS-ADOBE 2.0 Postscript format, that will be inserted at the top of every laser file. It could contain a company logo, standard header for a project, etc. It is inserted verbatim. For example you might have the **`.oa_pref`** line:

```
d3plot*laser_insert_file: /usr/local/logos/company_logo.psc
```

No check is made for its validity or syntactical correctness - that is up to you to get right.

Global preferences.

From version 9.3 onwards global preferences that apply to all programs can be specified using **`"oasys"`** as the program name.

`oasys*<keyword>: <argument`

At present the following global preferences can be defined

Preference	Type	Description	Valid arguments	Default
file_names	<string>	Controls input filename syntax. LSTC = d3*, OASYS = job.ptf*	OASYS, LSTC	OASYS
maximise	<logical>	Maximise window when Program is started	TRUE, FALSE	FALSE
placement	<string>	Location for initial window on multi-screen display	LEFT, RIGHT, BOTTOM, TOP, LEFT_BOTTOM, LEFT_TOP, RIGHT_BOTTOM, RIGHT_TOP	<none>
start_in	<string>	Directory to start Program in		<none>

The following options affect the appearance and behaviour of the graphical user interface, left handed support, and the mouse

Preference	Type	Description	Valid arguments	Default
display_factor	<real>	Factor on display size (0.5-2.0)	0.5 - 2.0	1.0
display_brightness	<real>	Menu brightness (0.0-1.0)	0.0 - 1.0	1.0
display_saturation	<real>	Menu colour saturation (0.0-1.0)	0.0 - 1.0	1.0
font_size	<string>	Menu font size	SMALL, DEFAULT, LARGE	DEFAULT
font_type	<string>	Menu font typeface and strength	HELVETICA, HELVETICA-BOLD, TIMES, TIMES-BOLD, COURIER, COURIER-BOLD	HELVETICA
left_handed	<string>	Left handed switching of mouse and/or keyboard	NONE, MOUSE, KEYBOARD, ALL	NONE
zoom_factor	<real>	Zoom Factor for mouse wheel(0.01-0.2)	0.01 - 0.2	0.05

The following strings and values control laser plotting setup

Preference	Type	Description	Valid arguments	Default
laser_paper_size	<string>	Default paper size	US, A4	A4
laser_orientation	<string>	Default page orientation	Portrait, Landscape	Landscape
laser_top_margin	<real>	Top margin size in mm		10
laser_bottom_margin	<real>	Bottom margin size in mm		30
laser_left_margin	<real>	Left margin size in mm		20
laser_right_margin	<real>	Right margin size in mm		10

If a preference is defined twice using both **`"oasys*"`** and **`"d3plot*"`** then the **`"d3plot*"`** setting will override the global setting.

APPENDIX III CHANGED DEFAULTS THAT AFFECT APPEARANCE

Changes between V9.3 and V9.2

- Spotweld beam "blobs" are now oriented into the local axis system of the beam, making it easier to see how spotwelds have rotated. (There is no method of reverting to the previous behaviour.)

Changes between V9.2 and V9.0

Apart from the changes to the user interface the only significant change to graphics is to **SI** Shaded Image data plotting mode.

- Previously this only produced gouraud shaded (fuzzy banded) plots.
- In V92 the default is now to produce plots with solid contour bands (in effect **CT** mode plots with lighting).
- The original "fuzzy" plots are still available from the popup menu on the **SI** button itself.

Changes between V8.x and V9.0

In order to improve appearance at initial startup the following changes to settings have been made:

Initial plot mode	Was	**LINE**	now	**SHADED**
Overlay mode	Was	**FULL**	now	**FREE**
Overlay colour	Was	**WHITE**	now	**GREY**

All of these may be configured using the relevant "oa_pref" file settings, see above, vis:

```
d3plot*initial_plot_mode:   LINE / HIDDEN / SHADED
d3plot*overlay_mode:        OFF / FREE / ALL
d3plot*overlay_colour:      WHITE, ... etc
```

Changes between V7.x and V8.0

In order to improve the initial appearance of plots some defaults which affect the appearance of shaded plots have been changed:

BRIGHTNESS	Was	100%	now	90%
SHININESS	Was	70%	now	70%
AMBIENT	Was	25%	now	40%
SHADED_SAT	Was	100%	now	50%
Light position	Was	at observer	now	above right shoulder

The effect on lighting plots will be to make them darker, less brightly coloured, and obviously lit from a different position. To restore the previous defaults it will be necessary either to edit the "**.oa_pref**" file (see below), or to reset these values explicitly in the **PROPS** menu.

The changes required in the **.oa_pref** file to restore the original lighting attributes are:

```
d3plot*shaded_ambient:      25
d3plot*shaded_diffuse:      90
d3plot*shaded_shininess:    70
d3plot*shaded_saturation:
                            100
```

Next section

APPENDIX IV COMMAND LINE OPTIONS AND WINDOWS FILE ASSOCIATIONS

UNIX

Instead of starting Oasys D3PLOT using the Oasys command shell it is also possible to start D3PLOT from the command line. Starting D3PLOT from the command line offers a number of advantages.

- Faster start-up is possible by pre-selecting the device type.
- The input filename can be specified and opened automatically.
- Faster start-up is possible by pre-selecting the device type

Argument format:

<application name> (<arg 1>) … (<arg n>) (<input filename>)

Valid D3PLOT command-line arguments

Graphics device type	**-d=<*device type*>** eg **-d=default**	Valid device types are: **opengl** OpenGL (3D graphics) **x8** X-windows, 8 bit-planes **x24** X-windows,24 bit-planes **x** X8 if present, else X24 **default** OpenGL if present, then X8 if present, then X24
Flag to start with window maximised (full screen)	**-maximise**	(No argument)
Command file name Flag to exit when command file run complete if desired	**-cf=<*filename*>** **-exit**	A valid command file name (usually *.tcf) (No argument)
Checkpoint file to replay Number of lines to execute in checkpoint file	**-replay=<*filename*>** **-rlines=<*nnnn*>**	A valid D3PLOT ckeckpoint file (usually **cp_d3plot***nnnn*) Where <*nnnn*> is a positive integer
Alternate "start in" directory (redefines current working directory)	**-start_in=<*pathname*>**	A valid directory (eg c:\my_files, /data/my_files)
Input database filename. (The extra time history (**.xtf**) and contact force (**.ctf**) databasees are also opened if present.)	<**filename**> eg **run_1.ptf**	A valid input file type: **name.ptf** (Complete state file) **d3plot** (ditto)

Some examples for D3PLOT might be:

pathname/d3plot83.exe -d=opengl -maximise run_2.ptf (Use device OpenGL, full -screen, open file run_2.ptf)

Note that no spaces should be left in the syntax <arg>=<value>.

For example: "**-d = x8**" is illegal.

Correct syntax is: "**-d=x8**"

WINDOWS (PCs)

Command-line arguments on Windows

It is possible to define command-line arguments under Windows: either directly when running an application from a MS-DOS prompt, or by defining "action" arguments when configuring a shortcut as described below. However this is unusual, and it is suggested that you seek advice from Oasys if you are not sure how to do this.

To make `.ptf` files open in D3PLOT by double-clicking on them

Under Windows on PC platforms it is possible to set up file associations so that double clicking on files with the **`.ptf`**, extension opens them automatically in D3PLOT. It is also possible to set up shortcuts so that double-clicking on the D3PLOT icon automatically starts the desired graphics mode.

All of these settings are optional: you should be aware that under the Windows operating system associating a filetype (via its extension) with an application is convenient, but can also be restricting and complicated to undo.

(1) If `.ptf` files do ***NOT*** currently have an application associated with them.

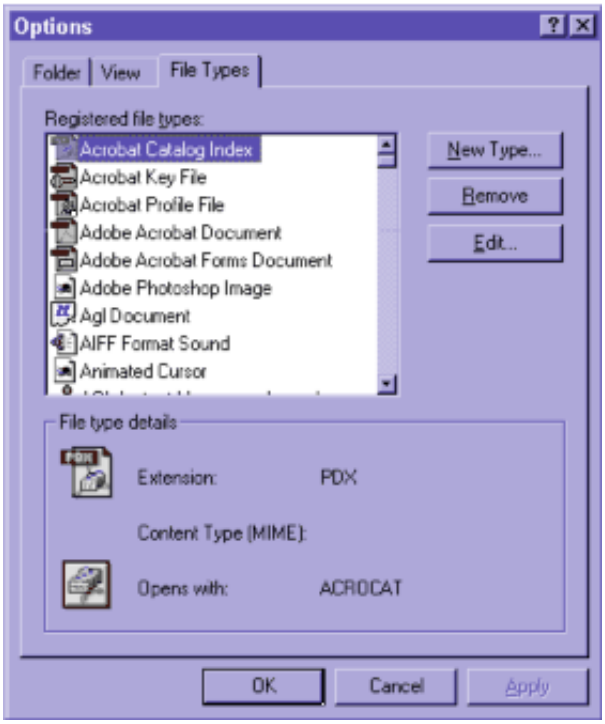
You can tell if no action is associated because a "double-click" doesn't work, and they have some non-specific, usually "windows", icon.

(If they ***do*** have an associated action already defined, go to here instead).

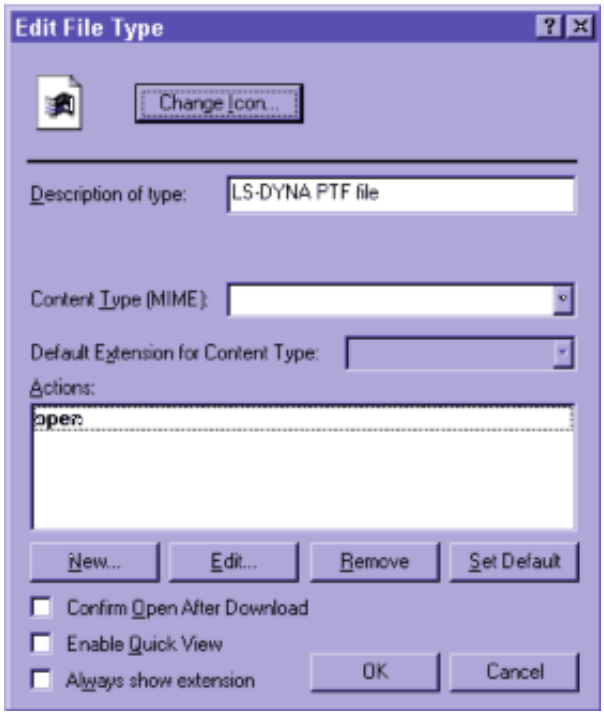
- Right click on any **`.ptf`** file, and select **Open with…** from the popup menu
- You will get the "**Open with**" panel.
- Type in a suitable description of the **`.ptf`** files. For example: "LS-DYNA complete state file"
- Make sure that the **Always use…** box here is ticked,
- Use the directory browsing window to find the correct D3PLOT executable. You are looking for file **`d3plot83.exe`**. (If you can't remember where it is installed use **Start-> Find->Files** or **Folders** to locate it.)
- Click on **OK** to close the "**Open With**" window. This should start D3PLOT, but at present it won't "know" about device type or filename: so select **STOP** to exit it.

If D3PLOT doesn't start check through what you've done, and ask Oasys for help if you can't resolve the problem.

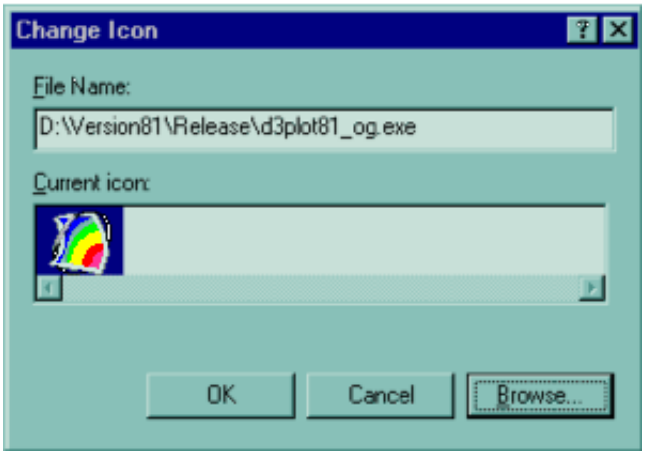
- In any folder window click on **View**, then select **Folder Options…** from the menu.



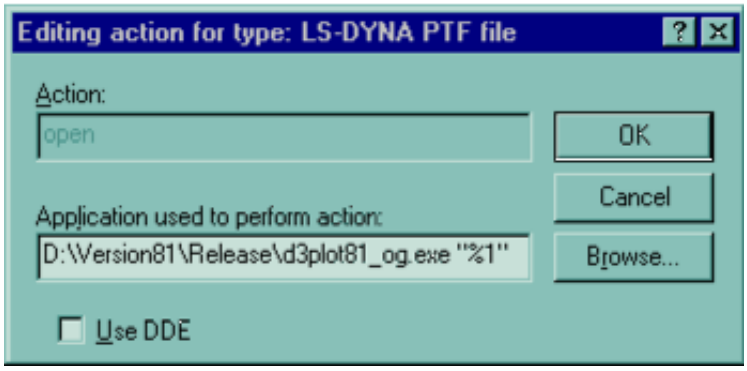
- Select the **File Types tab** at the top.
- Scroll down the list of Registered file types until you get to the name you entered for `.ptf` files e.g. "LS-DYNA complete state file". Highlight this and select **Edit…**



- In the Edit File Type panel click on **Change Icon…**
- And use **BROWSE…** in the Change Icon panel to find file **d3plot81_og.exe**.



- Then, in the Edit File Type panel, click on **Edit…** to obtain the Editing action panel:



The **Action:** entry should be set to "open" (greyed out).

You now need to edit the contents of the **Application used...** box:

At this stage you must decide what graphics driver you wish D3PLOT to open under. The options are:

opengl OpenGL 3D graphics. If your hardware supports it then use this, as it will give the fastest animation and best shading.

x24 X-Windows 2D graphics using 24 bit-planes. This will give good shading and lighting, but 24 bit-plane animation is slower than 8 bit-plane

x8 X-Windows 2D graphics using 8 bit-planes. This gives poorer quality shading and lighting, but faster animation than the x24 option.

default Chooses the best available option from the above.

- Edit the existing string to remove the "**%1**" at its end and replace it with **-d=<option>** where **<option>** is one of the above. For example:

  -d=opengl Selects OpenGL

  -d=default Lets D3PLOT choose the best option

It is recommended that you use:

d=default Which will select X, and provide for any future options.

(Note that there must be no spaces in this definition: **-d = x24** would cause an error. Also you must use lower case throughout. **-d=X24** would also cause an error.)

- Click on **OK** in the Editing action window to close and save it.
- Click on **Close** in the edit File Type window to close and save it.
- Click on **Close** in the Folder Options window to close and save it.

You should now find that:

- All **.ptf** files on your system show the D3PLOT icon.
- Double-clicking on any such file starts D3PLOT with the appropriate graphics driver, and opens that file.

It is not possible to set up the filename "d3plot" for double-clicking in this way since Windows requires filename extensions when assigning applications to files.)

(2) If **.ptf** files *DO* currently have an application associated with them

- In any folder window click on **View**, then select **Folder Options…** from the menu.
- Select the **File Types** tab at the top.
- Scroll down until you find the file type currently associated with **.ptf** files.
- Highlight it and click on **Edit…**

The procedure is then the same as the above, except that you are replacing…

- Icon (click on **Change Icon**)
- Application (click on **Edit…**)

...instead of defining new values.

If you are unsure it may be easier to **Remove** the current registered file type, then start afresh from (1) above.

APPENDIX V ENVIRONMENT VARIABLES USED BY D3PLOT

Environment variables can be used to set certain key parameters and to alter the default behaviour of the code. Generally these settings will be inserted into the "**oasys_xx**" command shell, but individual users are free to define their own parameters.

The setting of environment variables is done as follows:

Unix/Linux systems running "C" shell (/bin/csh) or its derivatives such as /bin/tcsh:

The format of the command is:

setenv <*parameter*> <*argument list*>

For example:

```
setenv DISPLAY my_machine:0
setenv SM_USE_VISUAL default
setenv DISPLAY_FACTOR 1.2
```

(Note that the "**oasys_xx**" shell is written using C shell syntax, so if it is amended the format above should be used.)

Unix/Linux systems running "Bourne" (/bin/sh) or "Korn" (/bin/ksh) shells

The format of the command is:

<*parameter*>=<*argument list*>; export <*parameter*>

For example:

```
DISPLAY=my_machine:0; export DISPLAY
SM_USE_VISUAL=default; export SM_USE_VISUAL
DISPLAY_FACTOR=1.2; export DISPLAY_FACTOR
```

Windows systems

Choose the "**System**" panel

==> ==> ==> **NT4**: Click on "Environment" tab

Win2K: Click on the "Advanced" tab, then "Environment Variables..."

Control Panel

System

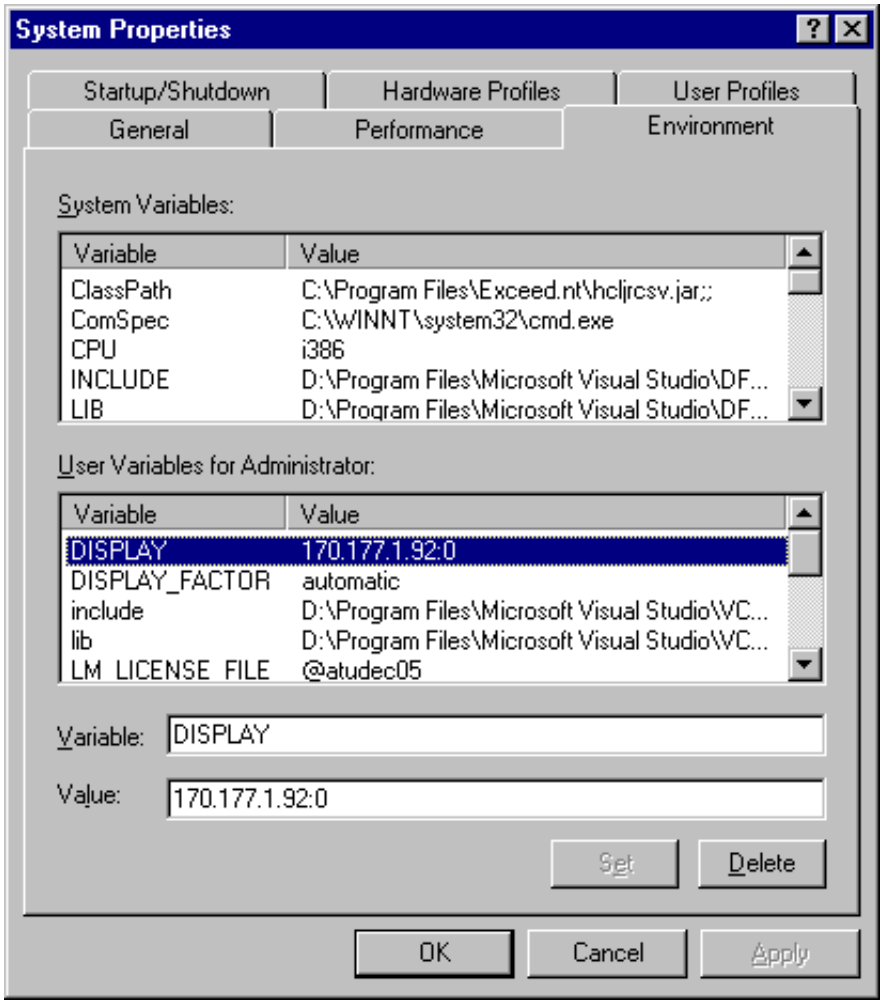
Alternatively right-click on on "**My computer**"

==> **NT4**: Click on "Environment" tab

Win2K: Click on the "Advanced" tab, then "Environment Variables..."

Right Click, select "Properties"

This panel is from a Windows NT4 machine, but the panel on Win2K or XP is very similar.



Then insert the relevant **Variable** and **Value** strings into the User or System settings as desired.

In this example it can be seen that iser Administrator has set the **DISPLAY** environment variable to **170.177.1.92:0**.

The following environment variables may be used to control the behaviour of D3PLOT.

Variable name	Description	Possible Values	Default
The following variables control the graphics and attributes of the display window and menu system.			
DISPLAY	The X11 display id on which graphics will be drawn. (This is ignored on "native" Windows systems.) If this is not defined (most systems initialise this to "**:0**") then no connection can be made to an X server, and no graphics will be drawn.	(<*machine name*>)**:<*server id*>**(<.*screen id*>)	**:0**
SM_USE_VISUAL	Sets the X11 "visual" id to be used for screen menus. Where a graphics display provides "overlay" planes these should normally be used, otherwise this should be left undefined or set to "default". Using an explicit visual id is possible, and this should be defined in hexadecimal (eg 0xf16). Experience has shown the on some Silicon Graphics systems using the "overlay" planes can result in very strange colours in other windows, in which case "default" should be used. Also on some W2000 and graphics board combinations problems may also arise with overlay planes and, again, "default" should be used.	**overlay** **default** <*visual id*> in hex	**overlay**
The following options may be used to configure the menu interface, and are intended for use by those who are having difficulties with the standard settings, or who have displays with unusual attributes. If you need advice about configuring your machine, and particularly if you suffer from physical impairment that may be eased by changing default settings, please contact Oasys and we will do our best to help.			
DISPLAY_SATURATION DISPLAY_BRIGHTNESS DISPLAY_FACTOR	Saturation controls the colour saturation (intensity) of menus Brightness controls the colour brightness of menus "Factor" sets the relative display scale, and can range from 0.5 (making menus larger) to 2.0 (making them smaller). It may also be set to "**automatic**" which derives a factor from the physical screen dimensions.	0.0 to 1.0 0.0 to 1.0 0.5 to 2.0, or **automatic**	**1.0** **1.0** **1.0**

SM_FONT_SIZE	Sets the size of fonts used in the menu interface. Possible values are: **SMALL** Will reduce the size of text in windows and buttons, and may be suitable on very large displays. **MEDIUM** "Normal" text size suitable for most purposes. **LARGE** "Large" text, which may be more easily read by those with visual impairment. **DEFAULT** The default size (usually medium) on this hardware.	**SMALL** **MEDIUM** **LARGE** **DEFAULT**	*<none>* (implicitly **DEFAULT**)
SM_FONT_TYPE	Sets the font (type-face) to be used for menu fonts HELVETICA / **HELVETICA-BOLD** Uses the proportionally spaced "Helvetica" (on X11) or "Arial" (on Windows) fonts. TIMES / **TIMES-BOLD** Uses the proportionally spaced "Times" font. COURIER / **COURIER-BOLD** Uses the fixed spaced "Courier" font. The "Bold" options for the font give a bolder and thicker font. The actual type-face used on a given platform will depend upon what fonts have been installed on that machine, and it is possible that the font and size combination selected may not be available. For advice on installing fonts please contact Oasys.	**HELVETICA** **HELVETICA-BOLD** **TIMES** **TIMES-BOLD** **COURIER** **COURIER-BOLD**	**HELVETICA**

LEFT_HANDED	Configures the keyboard and/or mouse for left-handed users. **KEYBOARD** Swaps the left and right "meta" keys (<shift>and <ctrl>) for the purposes of dynamic viewing. ***Note**: At present (V9.0) this does not work on Windows platforms for technical reasons. We hope to fix this in the future.* **MOUSE** Swaps the left and right mouse buttons for the purposes of all mouse actions (pick, drag, dynamic viewing) **ALL** Both of the above	**KEYBOARD** **MOUSE** **ALL**	*<none>*
The following two variables apply on **Windows platforms only**, and should only be used if the menu system is clearly obtaining the wrong display size from the system, as evidenced by fonts and menus being very much the wrong size.			
DISPLAY_HEIGHT	Set an explicit display height in millimetres	<height in mm>	<none>
DISPLAY_WIDTH	Set an explicit display width in millimetres	<width in mm>	<none>
The following options can be used to solve fairly obscure memory and display problems on X11 platforms.			
USE_PIXMAPS	Controls whether or not the menus use "pixmaps" (off-screen memory) to produce smooth scrolling. Turning this off (false) will save memory, and may help memory problems on a display that has only limited memory available for the X server, but will give slightly jerky window scrolling.	**true** or **false**	**true**
SAVE_UNDER	This flag was introduced to fix a specific bug on Compaq Alpha OSF4.x operating systems. Normally the window manager requests a redraw of windows that have been updated, even when they are currently obscured by something else. However the OSF4 window manager series failed to do this, leading to "bare" patches underneath popup menus when these were unmapped. Setting this flag to **false** results in more redraws on these systems since it suppresses the default "save under" property of X11 windows, but it does at least prevent windows getting bare areas. Compaq have fixed the bug in OSF5, and possibly in later releases of OSF4.	**true** or **false**	**true**

ALPHA_PERMIT_BROKEN	Another Compaq problem with older graphics cards under OSF4.x was a crash the first time broken lines for undeformed geometry were drawn. To prevent this Compaq machines have this capability switched off. To enable undeformed geometry to use broken lines on these machines set this variable to **true**.	**true** or **false**	**false**
The following are normally used when running command files, or performing automatic post-processing in batch mode.			
MENU_AUTO_CONFIRM	This variable is often used when replaying command files which, when recorded, paused and asked the user to confirm things. (For example HELP and Warning messages.) If the variable is set (true) then these will not pause and will behave as if the user had pressed "OK" - meaning that command files can play back without user intervention.	**true** or **false**	**false**
FILE_EXIST_ACTION	Action to be taken when opening a file for output, and the file already exists. Normally you will be prompted for the action to be taken when a file selected for output already exists. However if this variable is set to **overwrite** or **append** then the relevant action will be taken automatically. This is generally used when playing automatic post-processing batch scripts.	*<none>* **overwrite** **append**	*<none>*
SUPPRESS_CHECKPOINT	Suppresses the reading and writing of checkpoint files. This is desireable in batch mode since it prevents spurious checkpoint files being read in and processed.	**true** or **false**	**false**
The following controls the display of on-line manual pages on Unix systems only. (Windows systems use the default web browser.)			
NETSTART	Command string to start Netscape on Unix/Linux hosts. This is used to fire up the Netscape browser in order to read manual pages from within D3PLOT.	Any valid Unix command string.	*<none>*
The following variables control the default behaviour of the database management system.			

<table>
<tr><td>D3PLOT_CACHE_DATA</td><td>Default setting of the "CACHE_DATA" database switch. This controls whether or not D3PLOT attempts to store all data read from disk in its own core image. This setting can be changed manually during execution.
<table><tr><td>full (was true prior to V8.3)</td><td>Stores all data read from disk in memory until the database caching memory limits are reached. This can use a lot of memory and is not recommended unless disk access is very slow.</td></tr><tr><td>scalar</td><td>Stores basic nodal coordinates and the current component’s "scalar" data only. A reasonable trade-off between speed and memory size.</td></tr><tr><td>off (was false prior to v8.3)</td><td>Only basic nodal coordinate data are stored. Saves memory but may make data-bearing plots slow to modify.</td></tr></table>Generally Unix systems are better at cacheing disk data in spare system memory than Windows systems, making "disk" rereads faster since they are actually copied from memory. This makes the "off" option viable under Unix - however you may need to experiment.</td><td>full
scalar
off

(Also, for backwards compatibility:

true
false</td><td>scalar</td></tr>
<tr><td>D3PLOT_SOFT_LIMIT</td><td>Controls the amout of memory (in MB) set aside for the "soft" database limit</td><td>1 to 2048 MB</td><td>60% of system memory</td></tr>
<tr><td>D3PLOT_HARD_LIMIT</td><td>Controls the amout of memory (in MB) set aside for the "hard" database limit</td><td>1 to 2048 MB</td><td>80% of system memory</td></tr>
<tr><td colspan="4">The following variables set special parameters for data file reading and management, and are not normally used.</td></tr>
<tr><td>D3PLOT_SOLID_SE</td><td>Controls whether or not D3PLOT calculates strain energy density for solid elements. Doing this makes the assumption that the elements have not entered the plastic strain regime, so it is not normally set.</td><td>true or false</td><td>false</td></tr>
<tr><td>D3PLOT_SOLID_8</td><td>Analyses containing 8 integration points worth of data for solid elements are processed (NINTSLD = 8 on *DATABASE EXTENT BINARY), but only the first integration point is normally considered. If this flag is set then limited support for all 8 integration points is provided.

This facility is only partially implemented and should not be relied on.</td><td>true or false</td><td>false</td></tr>
<tr><td>PTF_CONTIGUOUS
CTF_CONTIGUOUS</td><td>Some versions of LS-DYNA sometimes contravene the rules for starting a new family member, and write data contiguously across file boundaries when they should not. Setting these variables may enable such files to be read.</td><td>true or false</td><td>false</td></tr>
</table>

D3PLOT_LIST_CROSSED	Normally coincident solid elements and those with crossed faces are dealt with by issuing a summary warning that they exist. Set this variable to see a detailed listing of all such elements.	**true** or **false**	**false**
The following variables are provided for debugging purposes only, and should not normally be used.			
XSYNC	Runs the X server in "synchronised" (unbuffered) mode. This will give woefully slow graphics, and is used for debugging purposes only.	**true** or **false**	**false**
WARN_REDEFINE	Makes the menu system issue a warning if a button is redefined. Again this is normally only used for debugging purposes.	**true** or **false**	**false**
CP_FILE_FILTER	When replaying checkpoint files this maps the file filter box and waits for user input, instead of using the path/filename encoded in the checkpoint file. This is used when replaying checkpoint files on a machine different to that on which they were written, or when the encoded file pathnames are no longer valid.	**true** or **false**	**false**
CP_DEBUG	When replaying checkpoint files this writes details of each command to <stdout>, and waits for <enter> before proceeding. **0** Off, the default. Checkpoint files play through without halts. **1** On. Checkpoint files echo all commands to <stdout>, and wait for a <return> on <stdin> before proceeding. However graphics commands (dynamic viewing, zoom, etc) play through without pause. **2** On. As for 1 above, but every command (including dynamic viewing) pauses and waits for confirmation before proceeding. Used for debugging problem files.	**0** (off) **1** **2**	**0**
CP_REFORMAT	When replaying checkpoint files written on a machine with a difference display resolution some picking and other screen-dependent operations may not work correctly. Setting this variable causes the playback machine to map the D3PLOT menu system onto a "virtual" display resolution equal to that of the original machine which *may* solve these problems if the two displays are not wildly different. It is more likely to work if the playback machine has a higher resolution than that on which the file was written.	**true** or **false**	**false**
D3PLOT_TIMING	Writes the time taken for each frame to be drawn to <stdout>. Used for timing comparison purposes.	**true** or <*none*>	**<***none***>**

APPENDIX VII DIALOGUE COMMAND SYNTAX

D3Plot has a dialogue command set that can be used in any of three ways:
1. In graphical (screen menu) mode commands can be typed into the "DialogueBox" at any time.
2. In non-graphical (text only) mode commands are typed in at the terminal prompt
3. In command files, run either interactively or in batch, commands areexecuted as if typed in.

In all cases the command ***input*** syntax is identical, although there are minor differences in output between "screen menu" and "text-only" modes: in the latter case all output has to go to the controlling terminal ("stdout"), whereas in the former separate windows are used for "help", "listing" and other output.

The Dialogue Command Structure

The command structure forms a hierarchical "tree", with the top-level **D3PLOT_MANAGER** at its "root".

The following rules apply:
- Command words may be abbreviated to any degree so long as:
  - they are unique in the context of their current menu
  - they must have at least their first two characters given

  For example **BP_BEAM_PLOTTING CT_CONTINUOUS** may be abbreviated to **BP CT**.
- Navigation up and down menu levels is performed as follows:
  - <command> takes you to the command's (sub-)menu level
  - Forward slash "/" takes you back to the top **D3PLOT_MANAGER** level before executing the following command(s)

  For example **BP_BEAM_PLOTTING** above takes you into the **BEAM_PLOTTING** sub-menu

  The command **/DEFORM EXPLODE** would work at the **BEAM_PLOTTING** prompt because it would return to the top level before parsing the **DEFORM** command.
- There is also a "global menu" of commands which is available at any (sub-)menu prompt.
  - These are primarily graphics commands that do not require a context
  - The commands can be listed with the **GM** (for Global Menu) command

- Any command can be aborted by typing **Q**(uit). This will return control to the next highest command prompt in the "tree"
- At any prompt you can type **H**(elp) to receive advice about what to do next.

Main Menu Commands:

CT_CONTINUOUS_TONE	Continuous tone (solid) contour plot
LC_LINE_CONTOURS	Line contour plot
CL_CLOUD_PLOT	"Cloud" (points) plot
ISO_SURFACE_PLOT	Iso-surface contour plot
VELOCITY_PLOT	Velocity arrows plot
CRITERION_PLOT	Criterion mode plot

The syntax is:

COMPONENT <data>	Defines the data component <data> to plot

ATTRIBUTE <attr> Defines the attributes of the criterion analysis. Valid <attr>s are:

LENGTH <length>	Symbol length
COLOUR <colour>	Symbol colour. Valid <colour>s are: **DATA_VALUE** Scales colours with data values **FIXED_COLOURS** Sets fixed colours
SYMBOL <symbol>	Symbol type. Valid <symbol>s are: **HIERARCHY** MAX stress has normal arrow head MID stress has flat T arrowhead MIN stress has inverted arrow head **ALL_LINES** All components drawn using plain lines

GO Perform the criterion plot

EXPLAIN Further help

STATUS Show the current settings

SI_SHADED_IMAGE Shaded image plotting options

The syntax is:

SHININESS <%age> Set the object shininess

SATURATION <%age> Set the object colour saturation

OVERLAY_COLOUR <colour> Set the overlay colour

LM_LIGHTING_MODEL <switch> Turn lighting model ON or OFF

BRIGHTNESS <%age> Set directional lighting intensity

AMBIENT_LIGHT_LEVEL <%age> Set ambient light level

SHADING_TYPE <option> Set smooth or flat shading. Valid <option>s are:

FLAT	Flat shading
SMOOTH <angle>	Smooth shading with an edge angle <angle>

DS_DITHER_SHADING <switch> Turn dithered shades ON or OFF

GOURAD_SHADING <option> Set solid or fuzzy contour bands. Valid <option>s are:

ON	Fuzzy bands
OFF	Solid bands

	DC_DITHER_CONTOURS <switch>	Turn contour dithering ON or OFF
	COLOUR_MAP	Draw the current colour map
	GO	Execute the plot
	STATUS	Show the current settings
	EXPLAIN	Further help
INTERFACE_ PLOTTING	Plot sliding interface results	
	The syntax is:	
	COMPONENT <data>	Define the <data> component to plot
	LC_LINE_CONTOURS	Line plot
	CT_CONTINUOUS_TONE	Continuous tone contour plot
	SI_SHADED_IMAGE	Shaded contour plot
	VECTOR_PLOT	Vector force plot
	HATCHING_SWITCH <switch>	<switch> hatching ON or OFF
	OPACITY_SWITCH <switch>	<switch> opacity ON or OFF
	EXPLAIN	Further help
BP_BEAM_PLOTTING	Plots beam data	
	The syntax is:	
	CT_CONTINUOUS	Continuous tone contour plot
	DP_DIAGRAM_PLOT	Bending moment diagram plot
	COMPONENT <data>	Defines the data component <data> to plot
	INTG_POINT	Set H-L beam extra dat integration point
	R2_REVERSE_END_2	Set the reverse end 2 switch on or off
	THICKNESS	Set the CT plot beam thickness
	ATTRIBUTES <attr>	Defines the attributes of the DP plot. Valid <attr>s are:

SIZE <size>	Size of the maximum diagram vector in screen units
HATCHING <hatch>	Intervals between the intermediate lines in screen units
PROJECTION <proj>	How the diagram is projected. Valid <proj>s are: **SCREEN** Always in the screen XY plane **LOCAL** In the local YY or ZZ plane
LABEL_VALUES	Set the label switch on or off

	OPACITY_SWITCH	Set the opacity switch on or off

	EXPLAIN	Further help
	WARNINGS	Some notes and caveats
	STATUS	Show the current settings
GREYSCALE	Draw a solid shaded plot	
	The syntax is:	
	COLOUR_OF_OBJECT <colour>	Set monochrome object colour
	SHININESS <%age>	Set object shininess level
	SATURATION <%age>	Set object colour saturation
	OVERLAY_COLOUR <colour>	Set overlay colour
	LM_LIGHTING_MODEL <switch>	Turn lighting model ON or OFF
	BRIGHTNESS <%age>	Set directional light intensity
	AMBIENT_LIGHT_LEVEL <%age>	Set ambient light level
	SHADING_TYPE <option>	Set smooth or flat shading. Valid <option>s are:

FLAT	Flat shading
SMOOTH <angle>	Smooth shading with an edge angle <angle>

	DS_DITHER_SHADING <switch>	Turn dithered shades ON or OFF
	GO	Execute the plot
	STATUS	Show the current settings
	EXPLAIN	Further help
COMPONENT <data>	Defines the data component <data> to plot	
STRESS_CONTROL	Control stress plotting	
	The syntax is:	
	PART_IGNORED_SW	Do / do not average across parts
	BLANKING_IGNORED_SW	Do / do not include blanked elements
	CLIPPING_IGNORED_SW	Do / do not include clipped elements
	AVERAGE_SWITCH	Do / do not average stress at nodes
	GLOBAL_COORDINATES	Use global coordinates
	LOCAL_COORDINATES	Rotate to local element coordinates
	SURFACE <surface>	Select a shell surface or layer. Valid <surface>s are:
		TOP, MIDDLE, BOTTOM, MAX_ALL, MIN_ALL, MAG_ALL or a layer number if present
	EXPLAIN	Further help

	STATUS	Show the current settings
CONTOUR	Control contour settings	
	The syntax is:	
	NUMBER_OF_LEVELS <levels>	Defines the number of contour levels
	AUTOMATIC	Autoscales the levels
	MANUAL_MAX_MIN <min> <max>	Manually sets the minimum <min> and maximum <max>
	USER_DEFINED	Define each contour line
	RESOLUTION <res>	Define the contour resolution. Valid <res>s are: LOW, MEDIUM or HIGH
	FORCE_LABELS	Force line labelling on LC plots
	LABEL_FREQUENCY <freq>	Set the labelling frequency on LC plots to <freq>
	LIMITING_VALUES <switch> <low> <high> <action>	Define what is contoured by upper and lower bound limits. Where: <switch> Is either ON or OFF <low> Is the lower limit <high> Is the upper limit <action> Is the exclusion behaviour and can be: OMIT, OUTLINE or BLACK
	OVERLAY_COLOUR <colour>	Set the overlay colour
	ARROW_LENGTH <length>	Set the length of arrows
	STATUS	Show the current settings
DISPLAY_OPTIONS	Set display parameters	
	The syntax is:	
	UNDEFORMED_SWITCH	Do / do not draw undeformed geometry
	BF_SWITCH	Do / do not draw back faces
	IF_SWITCH	Do / do not draw internal faces
	LOCAL_TRIAD_SWITCH	Do / do not draw triad
	CLOCK_SWITCH	Do / do not draw clock
	HEADER_SWITCH	Do / do not draw title header
	CONTOUR_BAR_SWITCH	Do / do not draw contour bar
	NASTRAN_CASES_SW	Do / do not draw nastran cases/freqs
	DATE_SWITCH	Do / do not draw date on plots
	BORDER_SWITCH	Do / do not draw border
	GRATICULE_SWITCH	Do / do not draw graticule

GRID_SWITCH Do / do not draw graticule grid

MODEL_BOX_SWITCH Do / do not draw box round model

ALL_NODES_SWITCH Do / do not draw all nodes

LABEL_SWITCH Control node / element labelling

ENTITY_SWITCH Control entity visibility

HO_HIDDEN_LINE_OPT <option> Set hidden line options. Valid <option>s are:

PAINTER	Use "painter" algorithm
RIGOROUS	Use "rigorous" algorithm
RESOLUTION <x> <y>	Set the resolution to <x> x <y>

FREE_FACE_OPTIONS <option> Set how free edges are displayed. Valid <option>s are:

OFF	Turn free edges off
ON	Turn free edges on
BE_BLANK_EDGES	Blanking does create free edges
BN_BLANK_NO_EDGES	Blanking does not create free edges
CE_CLIP_EDGES	Clipping does create free edges
CN_CLIP_NO_EDGES	Clipping does not create free edges
PE_PART_EDGES	Part boundaries create edges
PN_PART_NO_EDGES	Part boundaries do not create edges
SE_SURF_EDGES	Surface boundaries create edges
SN_SURF_NO_EDGES	Surface boundaries do not create edges

SEAT_BELT_OPTIONS <option> Set how seat belt elements are displayed. Valid <option>s are:

BELT_WIDTH <width>	Set the visual width
RETRACTOR_SIZE <size>	Set the visual retractor size
SLIP_RING_SIZE <size>	Set the visual slip ring size

SPRING_SYMBOL <option> Set how spring elements are displayed. Valid <option>s are:

ZIG_ZAG	Display springs as a zig-zag
LINE	Display springs as a line

BEAM_SYMBOL <option> Set how beam elements are displayed. Valid <option>s are:

LINE	Display beams as a thin line
THICK_LINE	Display beams as a thick line

	WINDOW <option>	Define the image window setting. Valid <option>s are:

FULL_SCREEN	The plot occupies the full screen
PART_SCREEN	The plot will not overwrite titleor contour key
REPORT_FORMAT	The colour hard-copied the image will be the same size as a Laser A4 plot
USER_DEFINED	Define the plot window with cursor

	STATUS	Show the current settings
VIEW_OPTIONS	Store and get views from a file	
	The syntax is:	
	STORE <view id> <name>	Store the current view in <view id> with the name <name>
	GET <view id>	Get the view <view id>
	DIRECTORY	List the stored views
	RENAME <view id> <name>	Rename a stored view <view id> to <name>
	DELETE <view id>	Delete view <view id>
	FILE_NAME <file_name>	Change the name of the file the views are stored in to <file_name>
	PERSPECTIVE <option>	Set perspective options. Valid <option>s are:

ON	Turn perspective on
OFF	Turn perspective off
DISTANCE <distance>	Set the eye to centre distance to <distance>

	WE_WRITE_EXPLICIT	Display the explicit centre and scale
	RE_READ_EXPLICIT <x, y, z> <scale>	Set the explicit centre to the coordinates <x y z> and the scale to <scale>
	STATUS	Show the current settings
	EXPLAIN	Further help
BLANK	Blank / unblank entities	
	The syntax is:	
	<entity> <list>	Blank the <list> of entities of type <entity>
	ON	Switch blanking on
	OFF	Switch blanking off
	ALL	Blank all entities except nodes
	UNBLANK <enitity> <list>	Unblank the list of entities of type <entity>
	REVERSE	Reverse the blanking
	STATUS	Show the current settings

UNBLANK Unblank entities

The syntax is:

<entity> <list> Unblank the <list> of entities of type <entity>

ALL Unblank all entities

VOLUME_CLIPPING Clip display by volume

The syntax is:

CREATE <option> Create a clipping volume. Valid <option>s are:

CARTESIAN <x_max, y_max, z_max> <x_min, y_min, z_min> <orient>	Define a box. Where: <x_max, y_max, z_max> and <x_min, y_min, z_min> — Define the corners of a box <orient> — Defines the orientation of the volume and can be: BASIC_MODEL, DEFORMED_MODEL or SCREEN_SPACE
CYLINDRICAL <x_c, y_c, z_c> <rad> <min_height> <max_height> <H_axis> <orient>	Define a cylinder Where: <x_c, y_c, z_c> — Defines the centre of the cylinder <rad> — Defines the radius of the cylinder <min_height> <max_height> — Define the height of the cylinder <H_axis> — Is the global axis to align the cylinder with and can be X, Y or Z <orient> — Defines the orientation of the volume and can be: BASIC_MODEL, DEFORMED_MODEL or SCREEN_SPACE

SPHERICAL <x_c, y_c, z_c> <rad> <orient>	Define a sphere Where: <x_c, y_c, z_c> Defines the centre of the sphere <rad> Defines the radius of the sphere <orient> Defines the orientation of the volume and can be: BASIC_MODEL, DEFORMED_MODEL or SCREEN_SPACE

	CANCEL	Cancels the clipping volume
	ORIENT	Defines the orientation of the volume and can be: BASIC_MODEL, DEFORMED_MODEL or SCREEN_SPACE
	DO_DISCARD_OUTSIDE	Remove elements outside the volume
	DI_DISCARD_INSIDE	Remove elements inside the volume
	OFF	Turn the clipping off
	ON	Turn the clipping on
	LOCATION_PLOT	Draw 4 views of the model to visualise the volume
	SKETCH	Sketch the volume on the current image
	STATUS	Show the current settings
	EXPLAIN	Further help
CUT_SECTIONS	Define a section that cuts through the model	
	The syntax is:	
	OFF	Turn the cut section off
	ON	Turn the cut section on

CREATE <option> Create a cut section. Valid <option>s are:

LS_DYNA_METHOD <x_nt, y_nt, z_nt> <x_nh, y_nh, z_nh> <x_eh, y_eh, z_eh> <orient>	Define normal and edge coordinates Where: <x_nt, y_nt, z_nt> Defines the Normal Tail coordinate <x_nh, y_nh, z_nh> Defines the Normal Head coordinate <x_eh, y_eh, z_eh> Defines the Edge Head coordinate <orient> Orientation of cutting plane and can be: BASIC_MODEL, DEFORMED_MODEL, SCREEN_SPACE
OV_ORIGIN_&_VECTORS <x_o, y_o, z_o> <x_x, y_x, z_x> <x_xy, y_xy, z_xy> <orient>	Define an origin, local x axis and local xy plane Where: <x_o, y_o, z_o> Defines the origin <x_x, y_x, z_x> Defines the local x axis <x_xy, y_xy, z_xy> Defines the local xy plane <orient> Orientation of cutting plane and can be: BASIC_MODEL, DEFORMED_MODEL, SCREEN_SPACE
N3_THREE_NODES <N1, N2, N3> <orient>	Define a plane with three nodes Where: <N1, N2, N3> Defines the three nodes <orient> Orientation of cutting plane and can be: BASIC_MODEL, DEFORMED_MODEL, SCREEN_SPACE

X_CONSTANT <x> <orient>	Define a plane of constant X Where: <x> Defines the x coordinate <orient> Orientation of cutting plane and can be: BASIC_MODEL, DEFORMED_MODEL, SCREEN_SPACE
Y_CONSTANT <y> <orient>	Define a plane of constant Y Where: <y> Defines the ycoordinate <orient> Orientation of cutting plane and can be: BASIC_MODEL, DEFORMED_MODEL, SCREEN_SPACE
Z_CONSTANT <z> <orient>	Define a plane of constant Z Where: <z> Defines the zcoordinate <orient> Orientation of cutting plane and can be: BASIC_MODEL, DEFORMED_MODEL, SCREEN_SPACE

CANCEL Cancels the cut section

SYSTEM <orient> Define the orientation of the cutting plane. Valid <orient>s are:

BASIC_MODEL	Tied to model space, undeformed geometry
DEFORMED_MODEL	Tied to model space, deformed geometry
SCREEN_SPACE	Tied to the screen

FSYS <system> Define the system for force and moment calculations. Valid <system>s are:

AUTOMATIC	Basic space will use global coordinates, deformed and screen space will use local coordinates
LOCAL	Local coordinates
GLOBAL	Global coordinates

POSITIVE_ACTION <action> Action for elements on positive side of the section. Valid <action>s are:

OMIT	Do not draw the elements
OUTLINE	Draw the elements in outline
NORMAL	Draw the elements in the display mode

NEGATIVE_ACTION <action> Action for elements on negative side of the section. Valid <action>s are the same as above

LOCATION_PLOT Draw 4 views of the model to visualise the section

VIEW_PLANE Change view to normal to plane

SKETCH Sketch the section

FORCE Calculate forces on current section

STORE <section id> <name> Store the current section in <section id> with the name <name>

GET <section id> Retrieve the stored section <section id>

RENAME <section id> <name> Rename a stored section <section id> to <name>

DELETE <section id> Delete section <section id>

DIRECTORY List stored sections on file

FILE_NAME <file_name> Change the name the sections are stored in to <file_name>

STATUS Show the current settings

EXPLAIN Further help

DEFORM Modify or deform the geometry

The syntax is:

EXPLODE <action> Separate parts. Valid <action>s are:

DEFINE <parts> <x, y, z>	Define the list of <parts> to explode along the vector <x, y, z>
CANCEL	Remove the vector

MAGNIFY_DISP <x, y, z> Magnify the displacements by <x, y, z>

FIX_NODE <action> Apply a negative translation, equal to the displacement of a node, to the whole model. Valid <action>s are:

DEFINE <node id>	Define the node <node id>
CANCEL	Remove the translation

SHIFT_DEFORMED <action> Apply a negative translation and rotation equal that defined by three nodes, to the whole model. Valid <action>s are:

DEFINE <N1, N2, N3>	Define the three nodes
CANCEL	Remove the translation and rotation

REF_NODE <action> Contour relative to a node or three nodes. Valid <action>s are:

SINGLE_NODE <node id>	Define the node <node id>
THREE_NODES <N1, N2, N3>	Define the three nodes
GLOBAL	Contour results in global system
LOCAL	Contour results in local system defined by nodes<N1, N2, N3>
REF_VALUES	Toggle on and off whether reference values should be used for WRITE and XY_DATA

REF_STATE <action> Contour relative to a state. Valid <action>s are:

ON	Turn reference state on
OFF	Turn reference state off
REF_MODEL <action>	Select a reference model. Valid <action>s are: **NUMBER <model id>** Set the reference model to <model id> **CURRENT** Set the reference model to the current model
SET_REF_STATE <action>	Select a reference state. Valid <action>s are: **NUMBER <state id>** Set the reference state to <state id> **TIME_RS <time>** Set the reference state to the state at time <time> **CURRENT** Use the current state
COORDS	Toggle on or off whether to apply reference to current coordinates
DATA_VALUES	Toggle on or off whether to apply reference to data values
UNDEFORMED	Toggle on or off whether to apply reference to undeformed geometry

STATUS Show the current settings

EXPLAIN Further help

EXPLODE <action> Separate parts. Valid <action>s are:

DEFINE <parts> <x, y, z>	Define the list of <parts> to explode along the vector <x, y, z>
CANCEL	Remove the vector

MAGNIFY_DISP <x, y, z> Magnify the displacements by <x, y, z>

FIX_NODE <action>

Apply a negative translation, equal to the displacement of a node, to the whole model. Valid <action>s are:

DEFINE <node id>	Define the node <node id>
CANCEL	Remove the translation

SHIFT_DEFORMED <action>

Apply a negative translation and rotation equal that defined by three nodes, to the whole model. Valid <action>s are:

DEFINE <N1, N2, N3>	Define the three nodes
CANCEL	Remove the translation and rotation

REF_NODE <action>

Contour relative to a node or three nodes. Valid <action>s are:

SINGLE_NODE <node id>	Define the node <node id>
THREE_NODES <N1, N2, N3>	Define the three nodes
GLOBAL	Contour results in global system
LOCAL	Contour results in local system defined by nodes<N1, N2, N3>
REF_VALUES	Toggle on and off whether reference values should be used for WRITE and XY_DATA

REF_STATE <action>

Contour relative to a state. Valid <action>s are:

ON	Turn reference state on
OFF	Turn reference state off
REF_MODEL <action>	Select a reference model. Valid <action>s are: **NUMBER <model id>** Set the reference model to <model id> **CURRENT** Set the reference model to the current model
SET_REF_STATE <action>	Select a reference state. Valid <action>s are: **NUMBER <state id>** Set the reference state to <state id> **TIME_RS <time>** Set the reference state to the state at time <time> **CURRENT** Use the current state
COORDS	Toggle on or off whether to apply reference to current coordinates
DATA_VALUES	Toggle on or off whether to apply reference to data values
UNDEFORMED	Toggle on or off whether to apply reference to undeformed geometry

	STATUS	Show the current settings
	EXPLAIN	Further help
WRITE	Write data to the terminal	
	The syntax is:	
	<entity> <list> <component>	Write <component> data for a <list> entities of type <entity>
	SCAN <entity> <nn> <component>	Display the maximum and minimum <nn> values of <component> for element type <entity>
	GS_GLOBAL_SUMMARY <summary>	Display a model summary. Valid <summary>s are:

GS_GLOBAL_SUMMARY	Whole model summary
PS_PART_SUMMARY	Whole model summary by part
NRB_NODAL_RB_SUMMARY	Nodal rigid body summary
IS_INTERFACE_SUMMARY	Contact surfaces summary

	COINCIDENT	Display coincident elements
	STATUS	Show the current settings
	EXPLAIN	Further help
XY_DATA	Plot XY Data	
	The syntax is:	
	NODES <list> <component>	Plot <component> data versus time for nodes in <list>
	SOLIDS <list> <component>	Plot <component> data versus time for solids in <list>
	BEAMS <list> <component>	Plot <component> data versus time for beams in <list>
	SHELLS <list> <component>	Plot <component> data versus time for shells in <list>
	THICK_SHELLS <list> <component>	Plot <component> data versus time for thick shells in <list>
	STONEWALLS <list> <component>	Plot <component> data versus time for stonewalls in <list>
	INTERFACES <list> <component>	Plot <component> data versus time for interfaces in <list>
	PARTS <list> <component>	Plot <component> data versus time for parts in <list>
	AIRBAGS <list> <component>	Plot <component> data versus time for airbags in <list>
	SURFACES <component>	Plot <component> data versus time for surfaces
	GLOBAL <component>	Plot global <component> data versus time
	COMP_XY <entity> <list> <component1> <component2>	Plot <component1> data versus <component2> over time for the <list> of <entity>s

COMP_LINE <entity> <list> <component1> <component2> Plot <component1> data versus <component2> at a given time for the <list> of <entity>s

SORT <action> Sort the COMP_LINE data points. Valid <action>s are:

NO_SORT	No sorting
LABEL	Sort by label ids
X_VALUE	Sort by data x value
Y_VALUE	Sort by data y value
BX_COORD	Sort by item basic x coordinate
BY_COORD	Sort by item basic y coordinate
BZ_COORD	Sort by item basic z coordinate
CX_COORD	Sort by item current x coordinate
CY_COORD	Sort by item current y coordinate
CZ_COORD	Sort by item current z coordinate

SET_INTERVALS <start> <interval> <finish> Set the start time for the first state to be extracted, the time interval between states and the time of the last state to be extracted

SELECT_STATES <list> Give a <list> of state numbers to be used

SHOW_TIMES Show state times in .ptf file

WRITE_CURVES Turn curve file writing on or off

CURVE_NAMES <type> Change the default curve file names. Valid <type>s are:

GLOBAL	Global data curves
PART	Part data curves
AIRBAG	Airbag data curves
CONTACT	Contact data curves
ELEMENT	Element data curves
NODE	Node data curves
COMPOSITE	Composite curves
SECTION	Cut section curves

NUMBER_OF_CURVES <curves> Set the maximum number of curves in a .cur file to <curves>

STATUS Show the current settings

EXPLAIN Further help

IMAGES Output images and animations

The syntax is:

JPEG <filename> Write the current image as a jpeg to <filename>

	BMP_U8 <filename>	Write the current image as an uncompressed 8-bit bitmap to <filename>
	BMP_C8 <filename>	Write the current image as a compressed 8-bit bitmap to <filename>
	BMP_U24 <filename>	Write the current image as an uncompressed 24-bit bitmap to <filename>
	PPM <filename>	Write the current image as a portable pixmap to <filename>
	PNG_8BIT <filename>	Write the current image as an 8-bit png to <filename>
	PNG_24BIT <filename>	Write the current image as a 24-bit png to <filename>
	GIF <filename>	Write the current image as a gif to <filename>
	MPEG <filename>	Write the current animation as an mpeg to <filename>
	AVI_MJPG <filename>	Write the current animation as a motion-jpeg to <filename>
	AVI_U8 <filename>	Write the current animation as an uncompressed 8-bit motion-jpeg to <filename>
	AVI_C8 <filename>	Write the current animation as a compressed 8-bit motion-jpeg to <filename>
	AVI_U24 <filename>	Write the current animation as an uncompressed 24-bit motion-jpeg to <filename>
	DITHER <level>	Set the dithering level
	FRAME_RATE <n>	Set to <n> number of frames per second for animations
	REPEAT <n>	Set to <n> number of repeats for animations
	QUALITY <%age>	Image quality of J/MPEG files
	STATUS	Show the current settings
LABEL	Display labels	
	The syntax is:	
	<entity> <list>	A <list> of <entity>s to label
	LABEL	Toggle on or off if entities are labelled with their node/element number
	PN_PART_NUMBER	Toggle on or off if entities are labelled with their part/surface number
	NE_NODES_ON_ELEMENT	Toggle on or off if nodes attached to an element are labelled
	EN_ELEMENTS_ON_NODE	Toggle on or off if elements attached to a node are labelled
	NC_NODAL_COORDINATES	Toggle on or off if nodes are labelled with their coordinates
	DATA_VALUE	Toggle on or off if nodes/elements are labelled with their current data value
	STATUS	Show the current settings
	EXPLAIN	Further help
UTILITIES	General utility commands	
	The syntax is:	
	COLOUR_OF_ENTITIES <entity> <list> <colour>	Set the colour of a <list> of <entity> type to <colour>

TRANSPARENCY <entity> <list> <%age>

Set the percentage transparency of a <list> of <entity> type to <%age>

MEASURE_DISTANCE <action>

Measure distances on screen. Valid <action>s are:

PP_POINT_TO_POINT	Distance between two screen points (cursor picked)
NN_NODE_TO_NODE	Distance between two nodes (cursor picked)
NO_NODE_TO_ORIGIN	Current node location (cursor picked)
NA_NODE_ANGLE	Angle between vectors n1n2 and n1n3 (cursor picked)
PA_POINT_ANGLE	Angle between vectors p1p2 and p1p3 (cursor picked)

TITLE_MODIFY <title> Change the model title to <title>

UNATTACHED_NODES List any unattached nodes

TARGET_MARKERS <action>

Add nodal target markers. Valid <action>s are:

RADIUS <rad>	Define the radius of the marker
COLOUR <col1> <col2>	Define the two colours <col1> <col2> of the marker
CREATE <node id>	Create a marker on node <node id>
DELETE <node id>	Delete the marker from node <node id>
OFF	Turn target markers off
ON	Turn target markers on

FAILURE_LOGIC <action>

How to handle failed elements. Valid <action>s are:

DS_DELETED_SWITCH	Toggle displaying deleted elements
DL_DELETED_LIST	List deleted elements
FH_HATCHING_SWITCH	Toggle hatching deleted elements
FC_FAILED_CONTOUR <colour>	Hatching colour
FL_FAILED_LIST	List failed elements
CRITERION	Define failure criterion

DATA_COMPONENTS List data components

CUSTOMISE_GRAPHICS <action> Graphic options. Valid <action>s are:

LW_LINE_WIDTH <pixels>	Define line width in pixels
M3D_3D_GRAPHICS	Switch to 3D mode
M2D_2D_GRAPHICS	Switch to 2D mode
UPDATE_LEVEL <level>	Define the update level
SOFT_CLIP_SWITCH	Turn the software clipping on or off
SP_SHOW_PROJECTION	Turn the box showing a representation of the projection and clipping planes on or off
WINDOW_SIZE <x> <y>	Set the window size

FILE_SKIP <n> Skips <n> missing .ptf files

FAMILY_SIZE <size> Set the file member size to <size> in MBytes

SETTINGS_FILE <action> Write or read a settings file. Valid <action>s are:

WRITE <filename>	Write to <filename>
READ <filename>	Read from <filename>

TOPAZ_FILE <To> <Tf> <Ti> <size> <header> <family> <filename> Write a topaz file

Where:

<To> is the time offset

<Tf> is the time factor

<Ti> is the time interval

<size> is the family member size in MBytes

<header> sets whether to write a header of initial data to file and can be YES or NO

<family> is the family member to start at

<filename> is the name of the file to write to

STL_FILE <filename> <type> Write an .stl file to <filename> as a file in the format <type>

Where:

<type> is either ASCII or BINARY

EXTERNAL <action> Plot externally defined data. Valid <action>s are:

READ_FILE <filename>	Read from <filename>
OFF	Switch data off
ON	Switch data on

GLOBAL_FACTORS <filename> Read <filename> for global factors

MENU_ATTRIBUTES Display the menu attributes panel

PTF_COMPRESS <action> Generate a new set of ptf files with a subset of the data. Valid <action>s are:

MODEL <model id>	Set the current model
FILENAME <filename>	Set the filename
PART <list>	Set a <list> of parts to output
STATES <list>	Set a <list> of states to output
APPLY	Write the ptf files
(NO_)VELOCITIES	Turn on/off output of nodal velocities
(NO_)ACCELERATIONS	Turn on/off output of nodal accelerations
(NO_)TEMPERATURES	Turn on/off output of nodal temperatures
(NO_)SHELL_STRESS	Turn on/off output of shell and thick shell stress tensor
(NO_)PLASTIC	Turn on/off output of plastic strain
(NO_)SHELL_FORCES	Turn on/off output of shell forces and moments
(NO_)SHELL_THICKNESS	Turn on/off output of shell thicknesses
(NO_)SHELL_EXTRA	Turn on/off output of shell extra variables
(NO_)SOLID_EXTRA	Turn on/off output of solid extra variables
(NO_)BEAM_EXTRA	Turn on/off output of beam extra variables
(NO_)STRAIN	Turn on/off output of strain tensor
(NO_)SPH_STRESS	Turn on/off output of strain tensor for SPH elements
(NO_)SPH_PLASTIC	Turn on/off output of plastic strain for SPH elements
(NO_)SPH_STRAIN	Turn on/off output of stress tensor for SPH elements

JAVASCRIPT Javascript interface

The syntax is:

COMPILE <filename> Compiles the javascript <filename> to check it for errors

EXECUTE <filename> Executes the most recently compiled javascript file or compiles and executes <filename> if given and different

MEMORY <size> Sets javascript memory arena to <size> in MBytes

STATUS Display programme status

NEW_MODEL <filename> Open model <filename>

EXIT Exit D3Plot

Global Menu Commands:

DRAW	Draws the undeformed geometry in line mode
LINE	Draws the current state in line mode
HIDDEN_LINE	Draws the current state in hidden-line mode
SHADED	Draws the current state in shaded mode
REDRAW	Redraws all windows
CW_CURRENT_WINDOW <window id>	Sets the current window to <window id>
CM_CURRENT_MODEL <model id>	Sets the current model to <model id>
SXY, SYZ, SZX	Preset views on XY, YZ and XZ axes
+XY, +YZ, +ZX	Alternative syntax for the above
-XY, -YZ, -ZX	Reversed views of the above
ISOMETRIC	Preset isometric view
+ISO, -ISO	Same as above and reverse
RS <x y z>	Rotate about screen coordinates <x y z>
RM <x y z>	Rotate about model coordinates <x y z>
ZM	Zoom in using cursor
CENTRE	Centre image on cursor position
MG <scale>	Magnify by factor <scale>
AU_AUTOSCALE_UNDEF	Autoscale on undeformed geometry
AC_AUTOSCALE_CURR	Autoscale on current geometry
ZERO_VIEW	Reset view to plan on XY plane
TIME <time>	Set to a state at time <time>
	Where:
	<time> is a time value, PREVIOUS, NEXT or LAST
STATE <number>	Set to a state by <number>
	Where:
	<number> is a state number, PREVIOUS, NEXT or LAST
FILE_SCAN	Re-scan file family for new states
SS_SHOW_STATES	Show all state times in the file
GM_GLOBAL_MENU	Display global menu command summary
GH_GLOBAL_HELP	Help on global commands
GE_GLOBAL_EXPLAIN	Details of global commands

APPENDIX VI JAVASCRIPT API

Description of D3PLOT API functions and methods.

The following pages describe the functions ("methods") available in the D3PLOT Javascript interface. For information about how to run Javascript files see section 11.

Return value	Function name	Purpose
boolean	CreateWindow(model_list);	Create a new window containing 1 or more models in <model list>
boolean	DeleteWindow(window_list, (*dispose flag*))	Delete (a) window(s), optionally "disposing" of orphan models
boolean	SetWindowActive(window_id, active_flag)	Sets the "active" flag on the specified window(s)
integer	GetWindowMaxFrame(window_id)	Returns the highest frame in <window_id>
boolean	SetWindowFrame(window_id, frame_number)	Displays frame <frame_number> in the specified window(s)
integer	GetWindowFrame(window_id)	Returns the current frame of <window_id>
object	GetWindowModels(window_id)	Returns an object containing information about the model(s) in <window_id>
boolean	SetCurrentModel(model_id)	Sets <model_id> to be current for data "get" and "put" operations
boolean	SetCurrentState(state_id)	Sets <state_id> to be current for data "get" and "put" operations
integer	GetNumberOf(type, (*further args*))	General routine to return the quantity of many different things
boolean	QueryDataPresent(component, *(type)*)	Returns JS_TRUE if data <component> is present.
double	GetTime(*(state)*)	Return the analysis time of the current state, or of <state> if defined.
integer	GetLabel(type, item, (*state*))	Return the external label of internal <type/item>
integer	GetPid(type, item, (*state*))	Return the internal part id of internal <type/item>
integer	GetMid(type, item, *(layer), (state))*	Return the external material id of internal <type/item>
object	GetTopology(type, item, (*state*))	Return an object containing topology, #nodes and part id for internal <type/item>
object	GetElemsAtNode(node, type, *(state)*)	Return a list of elements of <type> at <node>
double or array	GetData(component, type, item, *(int_pnt), (extra), (fr_ref), (state), (dda)*)	Return scalar, vector or tensor data for data <component> of <type/item>
integer	CreateUbinComponent(name, item type, data type, if_existing)	Create a user-defined internal binary (UBIN) data component and return its "handle"
object	LocateUbinComponent(name)	Returns an object with the <handle> and other attributes of UBIN component <name>
integer	DeleteUbinComponent(handle)	Deletes UBIN component <handle>
double or array	GetUbinData(ubin handle, type, item, int_pnt, *(state)*)	Get data from UBIN component <handle> for <type/item>
boolean	PutUbinData(ubin handle, type, item, int_pnt, data, *(state)*)	Insert data for UBIN component <handle> for <type/item>
boolean	SetCutSection(window_id, attribute, value)	Defines cut-section attributes in <window_id>
object	GetCutSection(window_id, *(state_id), (model_id)*)	Retrieves cut-section attributes from <window_id>
object	GetCutForces(window_id, *(include blanked), (part_id), (state_id), (model_id)*)	Returns the forces, moments, centroid and area from the cut-section in <window_id>
boolean	DialogueInput(line_1, *(line_2), ... (line_n)*)	Executes 1 or more lines of command-line dialogue commands
boolean	DialogueInputNoEcho(line_1, *(line_2), ... (line_n)*)	As for DialogueInput(), but with no echo to dialogue box

Utility functions common with the PRIMER Javascript API Global Class

boolean	Print(arg)	Prints <arg> on the terminal (stdout), but does ***not*** add <carriage return> or <line feed>. Successive Print() calls will append to the line.
boolean	Println(arg);	Prints <arg> on the terminal followed by a line feed

boolean	Message(arg);	Prints <arg> in the dialogue box followed by a line feed
boolean	Warning(arg);	Prints warning message <arg> in the dialogue box followed by a line feed
boolean	Error(arg);	Prints error message <arg> in the dialogue box followed by a line feed
string	NumberToString(<number>, <width>);	Formats <number> (integer or float) to a string using the specificed <width>
boolean	Sleep(nsecs);	Sleeps (pauses execution) for <nsecs> seconds.
integer	System(arg);	Issues command <arg> to the operating system
boolean	Exit();	Exits from D3PLOT

In addition the following classes are common with the PRIMER API:

File	Handles file opening, closing and general i/o
Window	Handles creation and management of menu system windows
Widget	Handles primitives and processing in Window classes

Full documentation of the Oasys PRIMER Javascript API may be found under primer_js_api.html

Notes on programming:

All scripts and script runs are independent

Javascript variables and "current" settings are not "remembered" in any way across successive executions of scripts. Each script, including a second and subsequent execution of the same compiled script, is wholly independent; and default current model, window and other values are reset every time a script is executed.

Argument types:

Javascript is a very weakly typed language in which data can be thought of as "numbers", "strings", "arrays", "objects" and so on. However when describing a function it is useful to be more precise about the sort of argument expected or returned, and the following descriptions are used below.

`Integer`	A "whole number" which would qualify as an integer in languages such as Fortran or C.
`Double`	A "floating point number". Javascript does all floating point arithmetic in double precision, so there is no concept of a single precision "float".
`Boolean`	Either **`JS_TRUE`** or **`JS_FALSE`**. Typically this is the success/failure status outcome of function calls that do not return a data value. It does not translate to a "number", and should only be used for logical tests.
`String`	A string of one or more characters in "..". For example "this is a string".
<*type*> **`Array`**	An array of values of <*type*>. Javascript does not assign types to arrays, and their subscripts may be of mixed type, however in the context of this API arrays will be of a single type, eg **`integer`**, which will be specified.
`Object`	Javascript objects may be of any type, and members may be added at will. In the context of this API they will be classes, and the class members will be specified.
`Constant`	Will be a capitalised constant value from a defined list (eg **`DX`**, **`SOLID`**, etc), effectively an integer. All valid constants for this API are listed in the Table of Valid Constants below.

A common error is to pass an argument that is a **`String`** to a function below that expects a "number" of some sort, typically when data has been read from an external file and processed using string manipulation functions. This will generate a "wrong type of argument" error when the function executes.

The solution is to use one of the Javascript conversion functions **`Number(`***xxx***`)`**, **`ParseInt(`***xxx***`)`** or **`ParseFloat(`***xxx***`)`** to convert the string to a number.

Compulsory and optional arguments:

Many functions have optional arguments. These will always be the trailing arguments and they will be written

italicised and placed in brackets in the function description. For example:

```
integer DemoFunction(arg_1, arg_2, (arg_3), (arg_4))
```

<arg_1> and **<arg_2>** are compulsory.
<arg_3> and ***<arg_4>*** are optional and may be omitted.

However if an optional argument is to be specified then any other optional arguments that precede must also be supplied. For example in the example above if <arg_4> is to be specified then <arg_3> must also be supplied even if it has the default value of zero.

Unless specified otherwise below a value of zero can be used for any optional argument that has to be supplied but is to be ignored.

Return values:

All functions in this API return a value, although you are free to ignore this. As a general rule those that "**Get**Xxxx" something return an integer, double, array or object as a result, and others return the boolean value **JS_TRUE** or **JS_FALSE** to denote success or failure. Each function's return type is documented below, and will be one of:

Boolean	**JS_TRUE** or **JS_FALSE**
Integer	An integer value
Double	A floating point value
Array	An array of values, usually all of type **Integer** or **Double**
Object	A Javascript "object" that is a structure with members defined by name.

Execution errors and warnings

Errors and warnings from the Javascript "engine" itself are sent to <stdout>. If a script fails to compile, or generates errors during execution, you should examine the controlling terminal window, or the log file if output has been piped to that.

Errors and warnings during execution of the functions described below will result in messages being sent both to the D3PLOT dialogue window and to <stdout>. Most such errors will result in termination of the script execution with error status, although there are a few cases where "harmless" errors, meaning unlikely to corrupt anything, will generate warnings and execution will continue.

Notes on data abstraction and processing.

Adapting programming style to improve memory efficiency

It is perfectly possible to import all the data from one or more states into the Javascript arena and to process it there, however you should be aware that data storage in Javascript is quite "bloated" (for example all scalar values are stored as 8 byte words) and that you may hit memory problems if you try to import too much data. It is possible to increase the size of the Javascript arena, but this still has to be allocated from the machine's storage heap and you may ultimately hit the memory limits of your machine.

Javascript also allows you to create and extend arrays and objects at will, making it very tempting to write scripts that exploit this flexibility. This is fine so long as you don't attempt to store too much in this way, since it is a very wasteful of memory, but if you start using "create and extend" for large quantities of data you will find that you run out of memory quite quickly.

With this problem in mind this API has been written in a way that will - hopefully - encourage you to keep your bulk data storage inside D3PLOT proper, and only to import data as and when it is required for processing. This will result in faster execution and fewer memory-related problems.

User Defined Binary Components

As an incentive to use memory efficiently an unlimited number of "User Defined Binary Components" (referred to as UBIN) may be created. These are very similar to the existing D3PLOT user-defined components, and are processed in much the same way. They have the following attributes:

- UBIN components are created from the Javascript by CreateUbinComponent().
- Each component must be assigned to one of the categories: SOSH (Solid and Shell), BEAM or NODE.
- Each component must be one of the types SCALAR, VECTOR or TENSOR.
- Each UBIN component has its values supplied via Javascript "PutUbinData()" functions.
- Similarly the data may be re-imported into the Javascript via "GetUbinData()" calls.

D3PLOT manages UBIN data in such a way that memory consumption is minimised, writing it out to (binary ".ubd") disk files if necessary and re-importing it from these files if required. This process is transparent to both the interactive user and the Javascript programmer, and means that the amount of data that is created and stored in this way is limited only by the disk space available on the machine.

UBIN data is also saved to file when no longer needed, or when D3PLOT exits, meaning that it is saved as an additional dataset and thus is automatically available during subsequent processing of a model.

The "current" model and state for data manipulation routines.

It is usually the case that you will be processing data for a single model and state at a time, and to save the need to specify these arguments to every data processing routine the data "put" and "get" routines in this interface operate by default on a "current state" in a "current model". The data "put" and "get" routines have an optional argument to specify a state different to the current one if required. Each model has its own, independent current state, and setting this will only affect the current model.

(Functions that process windows, or "by window" (eg cut-sections) use the window's model and state rather than this "current" one. Be careful of this distinction.)

Ordering data processing for efficiency: changing state number is expensive...

When you wish to process data over a range of states, for example to find a data envelope over time, you should bear in mind that because of the way storage is managed changing states is an expensive operation. Therefore it is better to perform all the processing in state A, then repeat for state B, and so on; rather that looping over all states separately for each item. For example:

GOOD: state loop is the outer one

```
for(istate=1; istate<=max_states;
istate++)
{
                    for(i=1;
                    i<=n_items; i++)
                    {
                              <get
                              data>
                              <process
                              it>
```

BAD: state loop is the inner one.

```
for(i=1; i<=n_items; i++)
{
                    for(istate=1;
                    istate<=max_states;
                    istate++)
                    {
                                                  <get
                                                  data>
                                                  <process
                                                  it>
```

Clearly there is also a trade-off to be made between local storage in the Javascript in the "good" example on the left versus slower speed in the "bad" one on the right.

... but the Direct Disk Access (dda) flag can be used if necessary.

Routine GetData() normally works on the assumption that you will want to read data for many items from the current state, therefore if the requested data is not currently in core it will read the complete data vector for all items of that type from disk into memory for that state, most likely re-using the memory used for the same data

vector in a previous state. This is efficient since all subsequent data reads for items in that state can be processed directly from memory without any further disk access, and it explains why changing states is a potentially expensive operation.

However the situation may arise where you want to read data for only a few items over a wide range of states, possibly in a random order, and the overhead of reading the complete data block for all items is prohibitively costly. In addition hopping back and forth between states might result in "churning" as data is read, discarded and reread repeatedly, making a bad situation worse.

The <dda> argument to GetData() will, if set to **ON**, change this behaviour so that complete data vectors are not read into memory, and instead data for the requested item **only** is read directly from disk, and then forgotten. This is an efficient solution when only a few items are being read from a large model over a range of states, but it will become progressively slower as more and more items are read, since each will require an explicit read from disk.

Clearly if data for enough items is read directly there will come a point where it is better to revert to the default behaviour. It is not possible to give guidance about where this point will lie since it will be a function of model size, number of states, computer memory capacity and speed of disk access; if you are writing a script that "hops about" states in a model you will have to experiment in order to find the best solution for your application. It is recommended that you try the default behaviour first (<dda> undefined or **OFF**), and only try setting it if the speed of your script is unacceptably slow.

Internal "item" numbers and external labels

Inside D3PLOT all nodes, elements, parts, etc are dealt with by their internal "item number" which is a sequence starting from 1 with no gaps, and external labels are used only when displaying data for the user; likewise element topology lists and lists of elements at nodes all refer to internal item numbers. The reason for this is obvious: finding data for an item number is a direct lookup, whereas an external label may be non-sequential and require a search to find its internal equivalent.

All the functions below which process data for explicit items take a pair of arguments (written as <type/item> for convenience):

Type code	One of the item type constants **NODE**, **SOLID**, etc
Item number	If **+ve** this is treated as an internal item number starting at 1 If **-ve** this is treated as an external label id.

Clearly it is far more efficient to use internal indices rather than labels, since the latter require a search to resolve them, however the option is available if required. When presenting data to the user, or writing it to file, the "GetLabel()" function can be used to return the external label of an internal <type/item> pair.

Ordering of data in vector and tensor arrays.

Where data is transferred in arrays the following data order is used.

Data type	Array length	Data order
Data vector (eg force vector, direction vector) Coordinate (eg origin, centroid)	array[3]	[X, Y, Z]
Data tensor (eg element stresses)	array[6]	[XX, YY, ZZ, XY, YZ, ZX]

To make this easier, and especially to avoid any ordering errors for tensor data, this API has the following constants defined:

- **X, Y, Z** for vectors
- **XX, YY, ZZ, XY, YZ, ZX** for tensors.
  (Tensors are symmetric, so constants **YX** (== XY), **ZY** (== YZ), **XZ** (== ZX) are also defined, it doesn't matter which are used.)

These are intended to be used for array subscripts to make coding clearer. For example in the following table both columns mean the same thing and are equally valid, but it is recommended that you use the left hand column's syntax as it is both clearer and less error-prone.

Using constants	... is much clearer than ...	Using numbers

`fx = a[X];`
`fy = a[Y];`
`fz = a[Z];`
		`fx = a[0];`
		`fy = a[1];`
		`fz = a[2];`

`sxx = b[XX];`
`sxy = b[XY];`
`szx = b[ZX];`
		`sxx = b[0];`
		`sxy = b[3];`
		`szx = b[5];`

Testing for the presence of a given data component.

One problem when post-processing data is that you cannot assume that a given data component will be present in a model database, as most output is switchable. The GetData() function will return values of zero for components that are not present, but will not issue any warning messages in the process. So in order to write robust scripts that will work with databases of unknown origin it is wise to use QueryDataPresent() to interrogate the database before attempting to extract data from components known to be optional.

Special considerations when working with adaptively remeshed analyses

When working with adaptively remeshed analyses you should bear in mind that each file family will almost certainly have a different number of nodes and elements, and that it is therefore extremely important to ensure that the item indices you are extracting are valid for the current state. To be on the safe side it is best to obtain the "number of" items every time you change state numbers.

In addition there is no guarantee that node or element <i> in family #1 will be the same in family #2. Exercise great care when extracting data from multiple families!

Notes on handling windows and models in windows.

The "current frame" in windows.

Windows in D3PLOT display results at a particular time in an analysis, each such time being a "frame", and animation displays the sequence of "frames". By default the times shown are those of each results state, meaning that "frame id" is equal to "state id". However this is not always the case, consider the following:

- The user has chosen to interpolate results by time, with the result that "frames" lie at fixed time intervals which no longer equate to states.
- The user is post-processing an eigenvalue analysis in which each "state" is in fact a given modeshape. In this case animation cycles a modeshape through the phase angles from -180 to +180 degrees in steps, with each phase angle step being a "frame".

Admittedly the two cases above are only rarely used, but the distinction between "frames" and "states" is important in these situations.

This API provides the following routines to deal with frames in window:

GetWindowMaxFrame(window_id)	Returns the highest frame in <window_id>
SetWindowFrame(window_id, frame_number)	Displays frame <frame_number> in the specified window(s)
GetWindowFrame(window_id)	Returns the current frame of <window_id>

Note that the "current frame" of a given window is purely an attribute of that window, and has no connection with the current model and state of this API as described above.

Models in Windows

D3PLOT requires an active window to contain at least one model, although it is possible to display any number of further models in the window. The following routines process and provide information about models in windows.

GetWindowModels(window_id)	Returns an object containing information about the model(s) in <window_id>

Processing windows containing multiple models can become difficult since each model's data at a given "frame" may contain results at a different time, and the attributes of each model may be different - for example not all models may contain a given data component.

Functions in this API which manipulate model data in windows may have an optional <model_id> argument to specify which model in a window is being processed. If this is omitted then the first model, as returned by function GetWindowModels(), will be used.

They may also have an optional <state_id> argument to specify the state number to be used. If this is omitted the model's state at the current "frame" in that window will be used.

It will be clear that processing data in windows containing multiple models is best avoided - please see "Recommended window setup when using this API" for suggestions about how to avoid these problems.

Recommended window setup when using this API

It will be clear from the above that departing from defaults of "one model per window" and "one frame per state" makes Javascript programming more difficult since the programmer has to add extra arguments to function calls to make sure that the correct data is being processed. Therefore it is ***strongly*** recommended that when scripts manipulate windows, or process data on a "per-window" basis (eg cut-sections), the following limitations should be adhered to:

- Each window processed should only contain a single model.
- The default of "show all states" should be used so that window "frames" are identical to model "states"

Detailed Description of Javascript Interface Functions.

Window manipulation

boolean **CreateWindow**(model_list)

Creates a new window containing one or more models contained in <model_list>.

The new window will always be the next free window, it is not possible to create windows "out of order" or to have gaps in the window numbering sequence.

<model_list> must be one of:

An array of model numbers. The length of this array is taken from its object property, or a model id of zero terminates the list.	At least one valid model number must be provided.
Asingle (scalar) model number	
The constant **ALL** to specify all models.	

At least one valid model number must be provided.

Arguments:	**<model_list>**	**integer array** or **integer** or **constant**	An array of model numbers or A scalar model number or **ALL**	Specifies the model(s) to be placed in the new window. At least one model number must be supplied.
Return value:	**<status>**	**Boolean**	**JS_TRUE** on success, **JS_FALSE** on failure.	

Example:

`a = new array(2, 3);` `CreateWindow(a);`	Create a new window containing models #2 and #3
`CreateWindow(6);`	Create a new window containing model #6
`CreateWindow(ALL);`	Create a new window containing all currently active models.

boolean **DeleteWindow**(window_list, *(dispose_flag)*)

Deletes one or more windows in <window_list>, dealing with "orphaned" models according to <dispose_flag>

<window_list> must be one of:

An array of window numbers. The length of this array is taken from its object property, or a window id of zero terminates the list.	At least one valid window number must be provided.
Asingle (scalar) window number	
The constant **ALL** to specify all windows.	

If a deleted window contains a model that is not in any other windows then that model becomes an "orphan".

By default orphan models are left in the database, but <dispose_flag>, if provided, controls this treatment.

WARNING:

- D3PLOT does not permit gaps in window numbering, therefore when a window is deleted any windows higher than this are renumbered downwards to fill the gap.
- However D3PLOT does ***not*** renumber models following the deletion of preceding ones. Deleted model ids simply become "inactive".

This means that following a window deletion operation:
- The total number of windows will change.
- Any window ids above those deleted will have been renumbered downwards.
- If any orphan models were deleted these models will now be inactive.
- If the current Javascript model has been deleted then the "current" model pointer will be reset to the first active model, or <undefined> if there are no such models.

Therefore if a script is to continue execution after a window deletion operation it is prudent to ensure that any "current" user-defined variables in the Javascript are reset to sensible values.

Arguments:	**<window_list>**	**integer array** or **integer** or **constant**	An array of model numbers or A scalar model number or **ALL**	Specifies the model(s) to be placed in the new window. At least one model number must be supplied.
	<dispose_flag>	**constant**	**LEAVE** or **DELETE**	**LEAVE** (default) leaves orphaned models in the database. **DELETE** deletes orphaned models.
Return value:	**<status>**	**Boolean**	**JS_TRUE** on success, **JS_FALSE** on failure.	

Example:

`a = new array(2, 3);` `DeleteWindow(a);`	Delete windows #2 and #3 leaving any orphaned models in the database.
`DeleteWindow(6, DELETE);`	Delete window #6, also deleting any orphaned models.
`DeleteWindow(ALL, LEAVE);`	Delete all windows, leaving any orphaned models in the database.

boolean **SetWindowActive**(window_id, active_flag)

Sets the "active" flag on a window.

When more than one window is in use it is convenient to be able to operate on a group of "active" windows with a single command in the Javascript, rather than having to loop over selected windows each time, and this function provides that capability. This activity status is used solely within the Javascript interface and does not have any bearing upon or connection with the Wn "tabs" used in the graphical user interface.

By default all windows are active (**ON**), but you can change this by setting the activity of specific windows **ON** or **OFF**.

Arguments:	**<window_id>**	**integer** or **constant**	Window number or **ALL**	Specifies the window(s) to have their status set
	<active_flag>	**constant**	**OFF** or **ON**	**OFF** makes the selected window(s) inactive **ON** makes window(s) active
Return value:	**<status>**	**Boolean**	**JS_TRUE** on success, **JS_FALSE** on failure.	

Example:

`SetWindowActive(1, OFF);`	Turns off the activity flag for window #1
`SetWindowActive(ALL, ON);`	Makes all current windows active.

integer **GetWindowMaxFrame**(window_id)

Returns the highest frame number in the specified window.

"Frame" number is usually the same as state number, but there are a few situations when this is not the case:

- Eigenvalue analyses. Each state is animated though <#frames> between +/-180 degrees phase angle
- Nastran-derived static analyses. Each loadcase is likewise animated through <#frames>
- Transient analyses that are being interpolated by time, giving (end time / time interval) frames.

In all cases animating a window results in it cycling through frames 1 to <max #frames>.

Arguments	**<window_id>**	**Integer**	Window number	Specifies the window number
Return value:	<**max #frame**>	**Integer**	Highest frame number in window	

Example:

`a = GetWindowMaxFrame(2);`	Get the highest frame number in Window #2

boolean **SetWindowFrame**(window_id, frame_number)

Sets the current "frame" in the window(s) specified to <frame_number>. The effect is immediate and the window(s) will be redrawn if necessary to show the requested frame.

See the notes in GetWindowMaxFrame() above on how frame number relates to state number.

Arguments:	**<window_id>**	**integer** or **constant**	Window number or **ALL**	Specifies the window(s) to have the frame number set
	<frame_number>	**integer**	The frame number to set	Should be a +ve integer value in the range 1 to max #frames in window. Values greater than max #frames are truncated to this
Return value:	**<status>**	**Boolean**	**JS_TRUE** on success, **JS_FALSE** on failure.	

Example:

`SetWindowFrame(1, 10);`	Set window #1 to display frame #10
`SetWindowActive(ALL, 3);`	Set all windows to display frame #3

integer **GetWindowFrame**(window_id)

Returns the current "frame" in window <window_id>

See the notes in GetWindowMaxFrame() above on how frame number relates to state number.

Arguments:	**<window_id>**	**integer** or **constant**	Window number (integer) or **ALL** (constant)	Specifies the window(s) to have the frame number set
Return value:	**<frame_number>**	**integer**	The current frame number in the specified window (integer)	

Example:

`a = GetWindowFrame(1);`	Get current frame of window #1

object **GetWindowModels**(window_id)

Returns information about the model(s) in window <window_id>, which must be a valid window number.

Every active window in D3PLOT must have at least one model, but may have any number.

Arguments:	**<window_id>**	**integer**	Window number (integer)	Specifies the window id
Return value:	**<model information>**	**object**	An object with the following fields:	
			.nm	The number of models in the window
			.list[ ]	An array of model numbers, of length <**nm**>
			If this function is called on an inactive window then <**nm**> will return zero, and <**list**> will not be defined.	

Example:

```
a = GetWindowModels(1);

for(i=0; i<a.nm; i++)
{
<do something with a.list[i]>
```

Get list of models in window #1

Setting "Current" status items

boolean **SetCurrentModel**(model_id)

Sets the current model for the Javascript interface to <model_id>.

At the start of script execution the current model is automatically set to the first active model in the database.

Arguments:	**<model_id>**	**Integer**	Model number	Specifies the model id to be made current
Return value:	**<status>**	**Boolean**	**JS_TRUE** on success, **JS_FALSE** on failure.	

Example:

`SetCurrentModel(2);`	Make model #2 current

boolean **SetCurrentState**(state_id)

Sets the "current" state for the Javascript interface to <state_id>

This is the state used for all "get" and "put" functions which handle model-related data. If the optional <state_id> argument in a get/put function call is used then that state is used instead for the duration of that call, but this current state is not changed.

The current state is a property of the current model, in other words each model has its own, separate, current state. For all models this defaults to state #1 (if present).

Setting the current state in model <i> has no effect on the current state in any other model.

Arguments:	**<state_id>**	**Integer**	State number	Specifies the state id to be made current
Return value:	**<status>**	**Boolean**	**JS_TRUE** on success, **JS_FALSE** on failure.	

Example:

SetCurrentState(27);	Make state #27 current

Functions to "get" and "put" data and other information.

integer **GetNumberOf**(type_code, (*state_id*))

Returns the quantity ("number of") items of type <type_code> in the current model.

Note that in adaptively remeshed models the current family may affect the number of nodes and elements returned. The family of the current state will be used unless you supply the optional <state_id> argument, in which case the family of that state will be used.

Arguments:	**<type_code>**	**Constant**	A valid type code or other constant from the list

Type code constant	Returns	Type code constant	Returns
WINDOW	Number of windows	**USER**	Total number of user-defined components
MODEL	Number of models	**UNOS**	Number of user-defined node scalar "
FAMILY	Number of families in current model (CM)	**UNOV**	.. node vector ..
STATE	Number of states in CM	**USSS**	.. solid/shell scalar ..
PART	.. Parts in CM ..	**USST**	.. solid/shell tensor ..
SURF	.. Contact surfaces in CM..	**UBMS**	.. beam scalar ..
		UBMV	.. beam vector ..
NODE	.. Nodes in Current Family (CF) ..		
SOLID	.. Solids in CF..	**NIP_H**	Number of Solid integration points
BEAM	.. Beams ..	**NIP_B**	Number of Beam int pts (integrated beams)
SHELL	.. Shells..	**NIP_S**	Number of Shell int pnts
TSHELL	.. Thick shells..	**NIP_T**	Number of Thick Shell int pnts
MASS	.. Lumped masses ..		
SPRING	.. Springs and dampers ..	**NEIPH**	Number of "extra" solid data values
SBELT	.. Seat-belts ..	**NEIPS**	Number of "extra" shell data values
RETR	.. Retractors..	**NEIPT**	Number of "extra" thick shell data values
SLIP	.. Sliprings ..		
PRET	.. Pretensioners ..		
JOINT	.. Joints ..		
WALL	.. Rigid walls ..		
SEGM	.. Contact segments ..		

	<state_id>	**Integer**	**Optional**: a state id (integer) which will be used instead of the current state. Only necessary in adaptively remeshed analyses.
Return value:	**<quantity>**	**Integer**	The number of items of this type.

Examples:

`a = GetNumberOf(MODEL);`	Return the number of models
`a = GetNumberOf(NIP_S);`	Return number of shell integration points
`a = GetNumberOf(SOLID, state_id);`	Return number of solid elements in family of state <state_id>

integer **QueryDataPresent**(component, *(type_code)*)

Returns **JS_TRUE** if data <component> is present in the current model's database, otherwise **JS_FALSE**.

For some data components that are switchable the <type_code> must also be supplied, these are listed below.

Arguments:	**<Component>**	**Constant**	A valid component constant from the list below.
	<type_code>	**Constant**	**Required for the following components:** **Component type** **Typical component constants** Stress tensor derived eg **SXX, ... SVON** Strain tensor derived eg **EXX, ... EVON** Effective plastic strain **EPL** Strain rate **ERATE** In the cases above one of the type codes **SOLID**, **SHELL** or **TSHELL** must also be supplied.
Return value:	**<True/False>**	**Boolean**	**JS_TRUE** if component is present **JS_FALSE** if not present

Examples:

if(QueryDataPresent(EPL, SOLID)) ...	Returns **JS_TRUE** if Effective Plastic Strain exists for solids
if(QueryDataPresent(TEMP)) ...	Returns **JS_TRUE** if nodal temperatures exist

integer **GetTime**(*(state_id)*)

Returns the analysis time of the current state, or that of <state_id> if defined.

Arguments:	***<state_id>***	**Integer**	**Optional**: State number	The state to be used instead of the current state.
Return value:	**<time>**	**Double**	The time of the state, or the frequency for eigenvalue analyses	

Example:

time = GetTime();	Returns the time of the current state.
time = GetTime(istate);	Returns the time of state <istate>.

integer **GetLabel**(type_code, item, *(state_id)*)

Returns the external label of internal <item> of type <type_code>.

Arguments:	**<type_code>**	**Constant**	A valid type code (**NODE**, **SOLID**, etc)	The type of the item
	<item>	**Integer**	The internal item number starting from 1	Its internal item number
	<state_id>	**Integer**	**Optional**: State number	The state to be used instead of the current state. Only necessary in adaptively remeshed analyses.
Return value:	**<label>**	**Integer**	The external label of the item	

Example:

a = GetLabel(NODE, 27);	Returns the external label of the 27th internal node.

integer **GetPid**(type_code, item, *(state_id)*)

Returns the **internal** part id of internal <item> of type <type_code>.

Arguments:	**<type_code>**	**Constant**	A valid part-based element type code (**SOLID**, etc)	The type of the item
	<item>	**Integer**	If +ve: The internal item number starting from 1 If -ve: The external label of the item	Internal item numbers will be many times faster to process
	<state_id>	**Integer**	**Optional**: State number (integer)	The state to be used instead of the current state. Only necessary in adaptively remeshed analyses.
Return value:	**<pid>**	**Integer**	The internal part id (internal, sequential numbering starting from 1)	

Example:

a = GetPid(SHELL, 27);	Returns the internal part id of the 27th internal shell.

integer **GetMid**(type_code, item, *(layer_id), (state_id)*)

Returns the **external** material id of internal <item> of type <type_code>.

Use of this function requires that material data be present, which means that a .ztf file must have been read.
If the optional <layer_id> argument is used the element must be in a part using a *PART_COMPOSITE definition.

If the material number is requested for a (composite) layer that does not exist in this item a value of zero is returned. No warning message is issued in this situation since experience has shown that this is a common occurrence and excessive warning messages are a nuisance.

Arguments:	**<type_code>**	**Constant**	**PART** or a valid part-based element type code (**SOLID**, etc)	The type of the item. If an element type then the part id is obtained from the element's topology list.
	<item>	**Integer**	If +ve: The internal item number starting from 1 If -ve: The external label of the item	Internal item numbers will be many times faster to process
	<layer_id>	**Integer**	**Optional:** For composites the layer number 1 - n	Ignored if omitted or zero
	<state_id>	**Integer**	**Optional**: State number (integer)	The state to be used instead of the current state. Only necessary in adaptively remeshed analyses.
Return value:	**<mid>**	**Integer**	The **external** material id	

Examples:

a = GetMid(PART, 2);	Returns the external material id of *PART 2
b = GetMid(PART, 12, 3);	Returns the external material id of the 3rd layer of *PART_COMPOSITE 12
c = GetMid(SHELL, 27);	Returns the external material id of the 27th internal shell.
d = GetMid(SHELL, 100, 2);	Returns the external material id of the second layer of internal shell 100. Assumes that the part of the shell is of type *PART_COMPOSITE

object **GetTopology**(type_code, item, *(state_id)*)

Returns the topology list for internal <item> of type <type_code>. This should only be used for element types which have nodal topologies.

Arguments:	**<type_code>**	**Constant**	A valid element type code (**SOLID**, etc)	The type of the item
	<item>	**Integer**	If +ve: The internal item number starting from 1 If -ve: The external label of the item	Internal item numbers will be many times faster to process
	<state_id>	**Integer**	**Optional**: State number	The state to be used instead of the current state. Only necessary in adaptively remeshed analyses.
Return value:	**<Topology>**	**Object**	An object with the following members:	

.nn	**Integer**	Number of nodes in topology list	
.top[ ]	**Integer array**	Array of internal node ids.	(Node and part ids are in internal, sequential numbering starting from 1.)
.pid	**Integer**	Internal part id if for part-based elements, otherwise zero.	

Example:

```
a = GetTopology(SHELL, 27);

nnodes = a.nn;
n1 = a.top[0];
n2 = a.top[1];
pid = a.pid;
```

Returns an object with the members described above..

object **GetElemsAtNode**(node, type_code, *(state_id)*)

Returns an object containing the number of elements of <type> at <node>, and also an array <list[ ]> of their internal indices.
If there are no elements of <type> at the node then JS_FALSE is returned.

Arguments:	**<node>**	**Integer**	If +ve an internal node index If -ve an external node label	The node at which to return the list of elements
	<type_code>	**Constant**	A valid element type code (**SOLID**, etc)	The type of the item
	<state_id>	**Integer**	**Optional**: State number	The state to be used instead of the current state. Only necessary in adaptively remeshed analyses.
Return value:	**<Element list>**	**Object**	If there are any elements of <type> at the node an object with the following members:	

.nn	**Integer**	Number of elements in <**list**[ ]>
.list[ ]	**Integer array**	Array of internal element indices

If there are no elements then the integer value **JS_FALSE**

Example:

`if(a = GetElemsAtNode(inode, SHELL))`
`{`
`nelems = a.nn;`
`e1 = a.list[0];`
`e2 = a.list[1];`
`}`
	Returns an object with the list of shell elements at node <inode>.
	
	If there are no elements the function returns **JS_FALSE**.

double *or* double array **GetData**(component, type_code, item, *(int_pnt), (extra), (fr_of_ref), (state_id)*, *(dda)*)

Returns the data for <component> of <item> of type <type_code>.

The return value is scalar, array[3] or array[6] for scalar, vector and tensor components respectively.

WARNING: If the function arguments are grammatically correct but the requested data component is not present in the database, then 1, 3 or 6 zeros are returned as required, ***and no warning message is output***. Therefore it is good practice to use function QueryDataPresent() to check that an optional data component is actually present in a database before attempting to extract its values.

<table>
<tr><td rowspan="8">Arguments:</td><td><component></td><td>Constant</td><td>A valid component code (eg DX, SXY)</td><td>Only valid codes in the list below are permitted.</td></tr>
<tr><td><type_code></td><td>Constant</td><td>A valid element type code (SOLID, etc)</td><td>The type of the item</td></tr>
<tr><td><item></td><td>Integer</td><td>If +ve: The internal item number starting from 1
If -ve: The external label of the item</td><td>Internal item numbers will be many times faster to process</td></tr>
<tr><td>int_pnt</td><td>Integer</td><td>Optional:
• If +ve is an integration point id (1 = lowest),
• Alternatively one of the codes TOP, MIDDLE, BOTTOM

Use zero to define a null "padding" argument</td><td>Integration points are only meaningful for some components, such as shell stress tensors and integrated beam stresses. This argument may be omitted if not needed.</td></tr>
<tr><td>extra</td><td>Integer</td><td>Optional:
• The "extra" solid or shell component id for components SOX or SHX
• The ALE multi-material group id for components AMMG and AMMS
• The sub-number for user-defined components UNOS, UNOV, USSS, USST, UBMS, UBMV

Use zero to define a null "padding" argument</td><td>This argument is only necessary for a few components, and may be omitted if not needed.</td></tr>
<tr><td>fr_of_ref</td><td>Integer</td><td>Optional: If supplied should be one of the constants:
• GLOBAL
• LOCAL
• CYLINDRICAL

Use zero to define a null "padding" argument</td><td>This argument is only necessary for directional components (eg X stress), and then only when something other than the default GLOBAL coordinate system is to be used.

If omitted, or set to zero, it defaults to GLOBAL for directional components, and is ignored for all others.</td></tr>
<tr><td>state_id</td><td>Integer</td><td>Optional: State number</td><td>The state to be used instead of the current state.</td></tr>
<tr><td>dda</td><td>Integer</td><td>Optional: "Direct Disk Access" flag.</td><td>Either OFF (default) for normal data cacheing
or ON to enable direct disk reading of data.

If turned on this reads data not currently in core memory directly from disk without loading the complete data vector for the state into core.

This should be used if you want to extract results for a few items over a range of states, since it will potentially be faster.</td></tr>
<tr><td colspan="4"></td></tr>
<tr><td>Return value:</td><td><Data></td><td>Double

or

Double array</td><td colspan="2">The return value will be one of three types:

double (scalar) — For data components that return a single scalar value (eg SXX)
double array[3] — For data components that return a vector value (eg UNOV)
double array[6] — For data components that return a tensor value (eg ETEN)

See table below for data component codes and return types.</td></tr>
</table>

Examples:	
`a = GetData(SXX, SHELL, 27, 2, 0, LOCAL);`	Returns the (scalar) X stress of internal shell #27 at integration point 2, in the element local coordinate system.

`b = GetData(ETEN, SOLID, 93);`
`sxx = b[0];`
`sxy = b[3];`
	Returns an array[6] of the strain tensor in solid element #93, implicitly in the global coordinate system.

`c = GetData(UNOV, NODE, inode, 0, 2, 0, 3);`
`vx = c[0];`
`vy = c[1];`
`vz = c[2];`
	Returns an array[3] of the 2nd user-defined Nodal Vector component at internal node #inode at state #3.

Functions for processing user-defined binary (UBIN) data components.

integer **CreateUbinComponent**(component_name, component_type, data_type, if_existing)

Creates a new user-defined binary (UBIN) component.

Note that user-defined components are "programme wide", so once created the data "slots" exist in all models. Data values that are not populated will return a value of zero.

Arguments:	**<component_name>**	**String**	A name for this component, character string up to 30 characters long.	The name must be unique, and it will be modified to make it so by appended numbers if an existing component of this name already exists.
	<component_type>	**Constant**	Must be one of the constants: • **U_NODE** for nodal data • **U_SOSH** for solid, shell & thick shell data • **U_BEAM** for beam data	User-defined components must fall into one of these three categories. It is not possible to have a component of a given name that contains data for more than one of these types.
	<data_type>	**Constant**	Must be one of the constants: • **U_SCALAR** for scalar data (any type) • **U_VECTOR** for vector data (**U_NODE** and **U_BEAM** only) • **U_TENSOR** for tensor data (**U_SOSH** only)	Choose the data type that matches the information you want to store.
	<if_existing>	**Constant**	Action to take if ubin component <component_name> already exists.	**REPLACE** deletes the existing ubin component, replacing it with this definition. This means that any existing data for the existing user-defined component of this name is deleted, and the component is re-initialised. **RENAME** changes the <component_name> argument of this function call by adding a suffix to make it unique, so that the existing component of this name (and its data) will be left unchanged, and the new one will not clash with it.

Return value:	**<Handle>**	**Integer**	A "handle" for the newly created component that should be used in subsequent "ubin" processing function calls. This "handle" should be regarded as private data, and not modified in any way. In addition if a UBIN component is created and then recreated and over-written in a script (**if_existing** = **REPLACE**) the "handle" from each call may be different - don't assume that it has not changed.

Example:

handle_1 = CreateUbinComponent("My nodal data", U_NODE, U_SCALAR);	Creates a component for nodal scalar data
handle_2 = CreateUbinComponent("My shell tensor data", U_SOSH, U_TENSOR);	Creates a tensor component for solid, shell and thick shell data

object **LocateUbinComponent**(component_name)

Locates an existing UBIN component <name> and returns its <handle>.

This is useful when a previous run has created a UBIN component and this script wishes to work with it. <name> is not case-sensitive, but an exact character match is required, so embedded white space is significant.

If the lookup succeeds this function returns the handle of the component (which will be a +ve integer), if it fails it returns the value JS_FALSE.

Arguments:	**<component_name>**	**String**	A name to search for, a character string up to 30 characters long.	Component names are not case-sensitive, but searching only succeeds if an exact match is found.

Return value:	**<Component info>**	**Object or JS_FALSE**	If successful this routine returns an object with members:	
			.handle	**Integer**	The "handle" of the UBIN component
			.ctype	**Constant**	One of **U_NODE, U_SOSH, U_BEAM**
			.dtype	**Constant**	One of **U_SCALAR, U_VECTOR, U_TENSOR**
			If unsuccessful it returns **JS_FALSE.**	

Example:

if(udata = LocateUbinComponent("My nodal data")) **{** **handle = udata.handle;** **...** **}** **else** **{** **... deal with failure**	Looks for component "My nodal data" and puts the result of a successful lookup in object <udata>.

integer **DeleteUbinComponent**(handle)

Deletes an existing UBIN component <handle>. The component is deleted from memory, and any ".ubd" files cached on disk are also deleted.

If this succeeds it returns JS_TRUE, otherwise JS_FALSE.

Arguments:	**<handle>**	**Integer**	The handle of an existing UBIN component
Return value:	**<status>**	**Integer**	**JS_TRUE** on success, **JS_FALSE** on failure.

Example:

if(!DeleteUbinComponent(handle_1)) **{** **...deal with failure...**	Deletes UBIN component <handle_1>

boolean **PutUbinData**(handle, item_type, item, int_pnt, data, *(state_id)*)

Stores data for <type/item> in the UBIN component <handle>.

This will overwrite any existing data in that "slot", which will be lost.

Arguments:	**<handle>**	**Integer**	A UBIN component handle as returned by CreateUbinComponent()	This must be a handle of an existing UBIN component.
	<item_type>	**Constant**	A constant of the standard type codes **NODE**, **SOLID**, **SHELL**, etc. This must match the underlying type of the UBIN component, thus **NODE** for components of type **U_NODE**, and so on.	It is illegal to attempt to store data for a type that does not match the underlying UBIN component type thus, for example, you cannot store **NODE** data for a **U_SOSH** component.
	<item>	**Integer**	If +ve: The internal item number starting from 1 If -ve: The external label of the item	Internal item numbers will be many times faster to process
	<int_pnt>	**Integer**	Integration point: must be a +ve layer number (lowest = 1) Or zero for item types **NODE** and **BEAM** that do not consider integration points in this context.	"Top", "Middle" and "Bottom" are not allowed in this context since "middle" is not writable in cases with an even number of points. A value of 1 should normally be used for solid elements
	<data>	Double or Double array	The data to be stored. Its format depends on the "data type" of the component:	The alignment of array members should be as follows: Vector: [X, Y, Z] Tensor: [XX, YY, ZZ, XY, YZ, ZX]
			"Data type" / **valid <data> type(s)** **U_SCALAR** / Scalar double, or double array[ ] of length >= 1. **U_VECTOR** / Double array[ ]of length >= 3 **U_TENSOR** / Double array[ ]of length >= 6	
	<state_id>	**Integer**	**Optional**: State number	The state to be used instead of the current state.
Return value:	**<status>**	**Boolean**	**JS_TRUE** on success, **JS_FALSE** on failure.	

Example:

```
dvec = new array(6);
dvec[XX] = sxx;
dvec[YZ] = syz;
PutUbinData(handle_1, SOLID,
27, 1, dvec);
```

Write an array of tensor data for solid #27, which implies that the UBIN data component <handle_1> is of type **U_SOSH**, and that its data type is **U_SCALAR**.

```
PutUbinData(handle_2, NODE, 17,
0, 19.5, istate);
```

Write the scalar value 19.5 for node #17, in state <istate>.

This implies that the UBIN component <handle_2> is of type **U_NODE** and its data is **U_SCALAR**.

double or double array **GetUbinData**(handle, item_type, item, int_pnt, *(state_id)*)

Retrieves data for <type/item> in the UBIN component <handle>.

If the data has not previously been written (a) value(s) of 0.0 will be returned.

Arguments:	**<handle>**	**Integer**	A UBIN component handle as returned by CreateUbinComponent()	This must be a handle of an existing UBIN component.
	<item_type>	**Constant**	A constant of the standard type codes **NODE**, **SOLID**, **SHELL**, etc. This must match the underlying type of the UBIN component, thus **NODE** for components of type **U_NODE**, and so on.	It is illegal to attempt to store data for a type that does not match the underlying UBIN component type thus, for example, you cannot store **NODE** data for a **U_SOSH** component.
	<item>	**Integer**	If +ve: The internal item number starting from 1 If -ve: The external label of the item	Internal item numbers will be many times faster to process
	<int_pnt>	**Integer**	Integration point: must be a +ve layer number (lowest = 1) Or zero for item types **NODE** and **BEAM** that do not consider integration points in this context.	"Top", "Middle" and "Bottom" are not allowed in this context since "middle" is not directly readable in cases with an even number of points. A value of 1 should normally be used for solid elements.
	<state_id>	**Integer**	**Optional**: State number	The state to be used instead of the current state.
Return value:	**<data>**	**double** **or** **double array**	Results are returned as: **"Data type"** / **Data type returned** **U_SCALAR** — Scalar double **U_VECTOR** — Double array[3] **U_TENSOR** — Double array[6]	

Examples:

`dvec = GetUbinData(handle_1, SOLID, 27, 1);`
`sxx = dvec[0];`
`szx = dvec[5];`
	Retrieve an array of tensor data for solid #27, which implies that the UBIN data component <handle_1> is of type **U_SOSH**, and that its data type is **U_SCALAR**.

`nval = GetUbinData(handle_2, NODE, 17, 0, istate);`	Retrieve the scalar value of node #17, in state <istate>. This implies that the UBIN component <handle_2> is of type **U_NODE** and its data is **U_SCALAR**.

Functions for processing cut-sections

boolean **SetCutSection**(window_id, attribute, value)

Sets an <attribute> of the cut_section in <window_id> to <value>.

Each D3PLOT window has a single cut-section which, by default, is not active. Its location, orientation and type can be defined here, and it can be turned on or off. Forces and moments from the cut-section can be obtained from function GetCutForces().

Cut section definitions are a "per window" attribute that apply to all models in the window. Thefore if the window has multiple models, and nodes are used to define the section (**N1** or **N3**), the origin and/or vectors of the section may vary for each model in the window. In addition if the coordinates of these nodes are "followed" (**FOLLOW_N**), then the section locations may change from state to state.

STATUS determines whether or not the cut section is active in the current window. The section does not have to be active in order to compute cut forces and moments.

SPACE determines whether the section is Lagrangian (**BASIC**) or Eulerian (**DEFORMED**). In the **BASIC** case the section is tied to the undeformed geometry and will move and distort as the model deforms, in the **DEFORMED** case the section is fixed in model space and the structure passes through it. For compatibility with LS_DYNA the way forces are calculated also varies with section space - see the documentation on GetCutForces() below.

The section can be defined by any of the following methods:

CONST_X **CONST_Y** **CONST_Z**	This is simplest. The plane will be aligned at a constant X, Y or Z axis depending on the **CONST_X/_Y/_Z** suffix, with its origin either at the explicit coordinate (array) <coord[3]>, or at the coordinate of node (integer) <node_id>.
OR_AND_V	An array of 9 numbers in three triplets: <origin coordinate>, <X axis vector>, <XY plane vector>. Local Z is obtained from X cross XY.
N3	An array of three integer node ids. Node #1 is the origin, N1N2 is the local X vector and N1N3 defines the local XY plane. Local Z is obtained from N1N2 cross N1N3
LS_DYNA	An array of 9 numbers in three triplets: <Normal tail coord (origin)>, <Normal head coord>, <X axis head coord>. Local Z and local X are obtained directly, and local Y from Z cross X.

FOLLOW_N(odes) determines whether or not a **DEFORMED** space section will track the motion of the node(s) used to define it. This is only meaningful for sections defined by:

- **N3.** The motion of all three nodes will update the origin and axes at each state
- **CONST_X/Y/Z** where a node has been used to define the origin. The node motion will update the section origin at each state, but its axes will remain constant.

<table>
<tr><td rowspan="11">Arguments:</td><td><window_id></td><td>Integer</td><td colspan="3">A valid window id, or ALL for all active windows.</td></tr>
<tr><td><attribute>
<value></td><td>Constant
Varies</td><td colspan="3"><attribute> / <value> pairs must be selected from the table below.</td></tr>
<tr><td colspan="3"><table>
<tr><th>Attribute</th><th>Type</th><th>Permissable values</th></tr>
<tr><td>STATUS</td><td>Constant</td><td>OFF or ON (default is OFF)</td></tr>
<tr><td>SPACE</td><td>Constant</td><td>BASIC, DEFORMED (default) (or SCREEN, not recommended)</td></tr>
<tr><td>OR_AND_V</td><td>Double array[9]</td><td><Origin> coordinate, <x axis vector>, <xy plane vector></td></tr>
<tr><td rowspan="3">CONST_X
CONST_Y
CONST_Z</td><td rowspan="3">Double array[3]
or
Integer node_id</td><td>Constant X plane centred at <origin> or at <node_id></td></tr>
<tr><td>Constant Y plane centred at <origin> or at <node_id></td></tr>
<tr><td>Constant Z plane centred at <origin> or at <node_id></td></tr>
<tr><td>N3</td><td>Integer array[3]</td><td>3 Node indices (+ve) or labels (-ve) to define the section</td></tr>
<tr><td>LS_DYNA</td><td>Double array[9]</td><td><Normal tail coord (origin)>, <Normal head coord>, <X axis head coord></td></tr>
<tr><td>FOLLOW_N</td><td>Constant</td><td>OFF or ON (default is OFF)</td></tr>
</table></td></tr>
<tr><td colspan="3">All coordinates and vectors must be defined in model space, and will always form an orthogonal right handed coordinate system in which local Z is normal to the cut plane.

Vector length is irrelevant (but should be well-conditioned), and the Y axis is obtained automatically from the vector cross product Z_AXIS x X_AXIS. If the Z and X axes as supplied are not at right angles the X will be updated to make it orthogonal to Y and Z.

The most recent of N1, N3 or ORIGIN will define the cut section origin coordinate.
The most recent of N3, or ORIGIN / X_AXIS / Z_AXIS will define the section orientation.

Care must be taken when defining nodes for windows that contain multiple models. Since a node index (+ve) may resolve to a different node in each model it is usually best to use external labels (-ve) in this context to avoid ambiguity. (The speed of the external => internal lookup will not matter as this function is unlikely to be called many times.)

FOLLOW_N(odes) will only have an effect if N1 or N3 were the most recently defined sources of origin and orientation.</td></tr>
<tr><td>Return value:</td><td><Boolean></td><td>JS_TRUE
or
JS_FALSE</td><td colspan="3">The return value will be JS_TRUE for success, or JS_FALSE for failure.</td></tr>
</table>

Examples:

SetCutSection(SPACE, DEFORMED);	Set the cut section to "deformed" (lagrangian) space.
var coord = new Array(1.0, 2.0, 3.0); **if(!SetCutSection(CONST_Z, coord))** **{** <deal with error>	Make the section constant in Z, with its origin at (1,2,3). Implicitly this means a plane of constant Z = 3.
var data = new Array(0.0, 0.0, 0.0, 10.0, 0.0, 0.0, 0.0, 10.0, 0.0); **if(!SetCutSection(LS_DYNA, data))** **{** <deal with error>	Make a cut-section using the LS_DYNA method with the origin at (0,0,0), Z axis pointing down the global X vector (head at (10,0,0)), and X axis down the global Y vector (0,10,0)

Object **GetCutSection**(window_id, *(state_id), (model_id)*)

Retrieves all attributes of the cut_section in <window_id>.

This routine returns a single object which is a structure containing all the attributes settable by SetCutSection() above.

Arguments:	**<Window_id>**	**Integer**	A valid window id.
	<state_id>	**Integer**	**Optional**: A valid state id. If omitted the state of the window's current frame will be used. This only matters if: • The section uses **N1** or **N3** definition methods ***and*** • It has been set to "follow_nodes In which case the section origin and/or vectors may change as the node(s) move.
	<model_id>	**Integer**	**Optional**: A valid model id that exists in <window_id>. If omitted the first model in the window will be used. This only matters if: • The section uses **N1** or **N3** definition methods ***and*** • The window contains more than one model. In which case the section origin and/or vectors in each model in the window may be different.

Return value:	**<Attributes>**	**Object**	The return value is a single object with the members:		
			Member name	**Type**	**Values returned**
			.status	**Integer**	Either **OFF** or **ON**
			.space	**Integer**	Either **BASIC, DEFORMED** or **SCREEN**
			.origin[3]	**Double array[3]**	Origin coordinates
			.x_axis[3]	**Double array[3]**	Local X axis vector (normalised)
			.y_axis[3]	**Double array[3]**	Local Y axis vector (normalised)
			.z_axis[3]	**Double array[3]**	Local Z axis vector (normalised)
			.definition	**Integer**	One of **OR_AND_V, CONST_X, CONST_Y, CONST_Z, N3, LS_DYNA** or zero if no section has been defined yet.
			.nodes[3]	**Integer array[3]**	For **definition == CONST_X/Y/Z**: nodes[0] = index of node if supplied. For **definition == N3**: nodes[0 to 2] = indices of three **N3** nodes. This array will always be present and have three entries, with unused entries being set to zero.
			.follow_n	**Integer**	**ON** if the section follows N1 or N3 as appropriate **OFF** if it does not. This value is only meaningful for **definition** == **CONST_X/Y/Z** where a <node_id> was supplied, and **N3**
			If the call fails the boolean value **JS_FALSE** is returned instead.		

Examples:

a = GetCutSection(1);	Retrieve the attributes of the cut section in window 1 at the current state and model.
a = GetCutSection(2, 10, 3);	Retrieve the attributes of the cut-section in window 2, at state #10 in model #3.

Object **GetCutForces**(window_id, *(include blanked), (part_id), (state_id)*, *(model_id)*)

Returns the forces, moments, centroid and area of the cut section in <window_id>.

The optional arguments allow further refinement of what is computed.

This routine returns an object with separate members for force, moment, centrod and area: see "return value" below for data format and coodinate system.

WARNING #1: Cut-sections in D3PLOT are a "per window" attribute, cutting all models in a window at the current "frame".

If the optional <state_id> argument is not supplied the forces and moments returned will be at the state of the current "frame" of the window, and while this will normally be the same as the current "state" this is not necessarily the case, since the user may have interpolated results by time.

Likewise if the optional <model_id> argument is not supplied then the model used will be the first in the window (as reported by function GetWindowModels()), which may not be the same as the "current model" of the Javascript interface.

Therefore to avoid ambiguity when extracting cut-section forces and moments it is recommended that:
- The window being used should only contain a single model
- The window should be set up to display all states without interpolation, thus <state id> == <frame id>. (This is the default for windows.)

This "single model in a window" approach is strongly recommended in this context since visual feedback will then match computed values.

WARNING #2: By default computed forces do NOT include blanked elements.

Since cut section display is primarily intended to be used interactively the default behaviour is to omit blanked elements from the force and moment calculation, since in this way the reported values match what is visible on the screen.

This behaviour is not ideal for batch processing since the user can, by manipulating blanking, change the results which are computed. Therefore the optional argument <include_blanked> may be used to override this behaviour and to force blanked elements to be considered. If omitted, or set to zero, then the default behaviour of omitting blanked elements will continue.

WARNING #3: Cutting a model exactly at mesh lines can result in ill-conditioned force and moment calculation.

It is tempting to define cut planes at nodes since this is easy to do, however this can give rise to ill-conditioning in a rectilinear mesh since the cut may lie exactly on the border between two adjacent elements and therefore won't "know" which one's results to use. Since LS-DYNA elements are constant stress there can be a step change in data values at element borders, and moving the cut plane by a tiny amount can result in a correspondingly large change in cut force and moment values.

It is strongly recommended that cut section definitions used for force and moment extraction should be located away from mesh lines so that they cut elements near their centres, thus avoiding any ambiguity about which elements to use.

Arguments:	**<Window_id>**	**Integer**	A valid window id.		
	<Include Blanked>	**Integer**	**Optional:**	**0**	To omit blanked elements (default)
				1	To include blanked elements
	<Part id>	**Integer**	**Optional:**	**0**	All part ids considered (default)
				<part_id>	Only forces in <part_id> will be computed. If +ve this is the internal part index If -ve this is the external part label
	<State_id>	**Integer**	**Optional**: A valid state id. If omitted the state of the window's current frame will be used.		
	<Model_id>	**Integer**	**Optional**: A valid model id that exists in <window_id>. If omitted the first model in the window will be used.		

Return value: **<Results>** **Object** The return value is an object with the following members

Member name	Data format	Content	Data alignment
.force[3]	**Double array[3]**	3 forces	[Fx, Fy, Fz]
.moment[3]	**Double array[3]**	3 moments	[Mxx, Myy, Mzz]
.centroid[3]	**Double array[3]**	Cut section centroid	[Cx, Cy, Cz]
.area	**Double**	Cut section area	[Area]

The coordinate system of these results depends upon the cut section's space system as follows:

BASIC space	Forces are moments are always returned in the **global** axis system, about the geometrical centre of the cut elements at the given state. Therefore the effective origin is likely to change as the model deforms. (This is the method used by LS-DYNA)
DEFORMED space	Forces and moments are returned in the **plane local** axis system, about the current section origin. The origin and axes will remain fixed as the model deforms unless one of the "section follows node(s)" options has been used.
SCREEN *space*	*Forces and moments are also returned in the **plane local** axis system. This space system is not suitable for computing results since these will change as the user updates the view, and therefore its use in this context is strongly deprecated.*

If the call fails the boolean value **JS_FALSE** is returned instead.

Examples:

**if(a = GetCutSection(1))
{
fx = a.force[X];
myy = a.moment[Y];
area = a.area;**
	Retrieve the results of the cut section in window 1 using all default attributes.

b = GetCutSection(2,0,3,12,1);	Retrieve the results from unblanked elements in part 3 using the section in window 2 for state 12 in model 1.

Functions for inserting command-line dialogue input

Boolean **DialogueInput**(line_1, *(line_2), ... (line_n)*)
Boolean **DialogueInputNoEcho**(line_1, *(line_2), ... (line_n)*)

Executes one or more command-line syntax commands <line_1>, *(<line_2> ... <line_n>)*. There is no limit to the number of lines that may be specified in a single call. See Dialogue Command Syntax for a full list of command-line commands

The **NoEcho** variant is identical, except that it suppresses the echo of the commands to the dialogue box.

D3PLOT provides a full command-line syntax as an alternative to graphical user interface commands, and a sequence of such commands may be provided here.

Note that:

- Each call to **DialogueInput** starts at the top of the D3PLOT command-line "tree", at the **D3PLOT_MANAGER >>>** prompt
- Each call is autonomous, there is no "memory" of where in the command-line tree previous commands finished.
- However within a single call the current command-line tree is remembered from one line to the next.
- Commands are not case-sensitive, although filenames and titles in command strings are.

Therefore commands which require more than one line of input to complete must be specified in a single call; and it makes sense to group a sequence of related commands together in a single call, although this is not mandatory.

If this succeeds it returns JS_TRUE, otherwise JS_FALSE.

Arguments:	**<line_1> ...**	**String**	A complete or partial command, as it would be typed in
	(<line_2>) and so on to ***(<line_n>)***	**String**	The next line, or the logical continuation of the previous line
Return value:	**<status>**	**Integer**	**JS_TRUE** on success, **JS_FALSE** on failure.

Examples:

DialogueInput("BLANK SOLID ALL", "UNBLANK SOLID 1 to 10", "HIDDEN");	Blanks all solids. Unblanks solids 1 to 10. Performs a hidden line plot. All commands are echoed to the dialogue box
DialogueInputNoEcho("STATE 10", "/GREY GO", "/IMAGE jpeg image.jpg");	Read state 10 Performed a shaded ("greyscale" in command-line syntax) plot Create a JPEG format file "image.jpg" Command is not echoed to the dialogue box.

Functions for diagnostic ouput

Boolean **Print(**arg_1, *(arg_2), (arg_3), ...*)

Prints argument(s) <arg_1> to <arg_n> to the controlling terminal (<stdout> in C). This is a crude function intended for diagnostic output only, and you have very little control over data formatting.

Note that:
- Arguments may be strings, or anything that can be converted to a string (eg a number).
- At least one argument must be supplied, but any number may be used. They can can mix strings and numbers at will.
- This function does ***not*** append a line feed to the end of output, to do this output the C language line feed character "\n" (see the example below).
- If you don't add a line feed then successive calls to <Print> will append output on the same line. However output is buffered, so you will only see a result on <stdout> when you eventually supply the <line feed> character to terminate the line. Building up a line of output from a succession of calls is a perfectly valid process.

If this succeeds it returns JS_TRUE, otherwise JS_FALSE.

Arguments:	**<arg_1> ...**	**String or number**	1 or more arguments that may be strings or numbers, the latter will be converted to strings.
Return value:	**<status>**	**Integer**	**JS_TRUE** on success, **JS_FALSE** on failure.

Examples:

`a.area = 3.14159;`
 	
`Print("The answer is " + a.area + "\n");`
	Prints the message
	
	"**The answer is 3.141592**"
	
	to <stdout>, terminating the line with a <line feed> (because the line ends "\n".).
	
	Note the use of the "+" operator which when used with strings in Javascript causes them to be concatenated into a single result. Note also that the number <a.area> is automatically converted into a string by Javascript in this context.

`Print(a.x, ", ", a.y, ", ", a.z, "\n");`	Prints the [x, y, z] components of object <a> in a comma-separated format, followed by a line feed.

List of valid constants to be used in the functions above:

General constants:		**Item type codes**	
OFF	Switch off	ELEM	Generic elements
ON	Switch on	NODE	Nodes
		SOLID	Solids
ALL	All of a category	BEAM	Beams
		SHELL	Shells
LEAVE	Leave behind (eg don't delete)	TSHELL	Thick shells
DELETE	Delete	MASS	Lumped masses
REPLACE	Replace	SPRING	Springs (discrete elements)
RENAME	Rename	SBENT	Seat belt types generally
		SBELT	Seat-belt elements
STATUS	Return status of something	RETR	Retractors
		SLIP	Slip-rings
WINDOW	D3PLOT window id	PRET	Pretensioners
		JOINT	Joints
MODEL	D3PLOT model id	WALL	Rigidwalls
FAMILY	Model family id		
STATE	Analysis state id	PART	Parts
		SURF	Contact surfaces
USER	User-defined component	SECT	(Element) section definitions
Cut-section constants		**Data extraction**	

SPACE	Space system call argument	TOP	Top shell surface
		MIDDLE	Middle shell surface
BASIC	Basic (eulerian) space	BOTTOM	Bottom shell surface
DEFORMED	Deformed (lagrangian) space		
SCREEN	Screen space	GLOBAL	Global coord system
		LOCAL	Element local system
OR_AND_V	Origin & vectors definition method	CYLINDRICAL	Cylindrical system
CONST_X	Constant X defn method		
CONST_Y	Constant Y defn method	NIP_H	#Solid intg points
CONST_Z	Constant Z defn method	NIP_B	#Beam intg points
N3	3 nodes defn method	NIP_S	#Shell intg points
LS_DYNA	ls-dyna defn method	NIP_T	#Tk shell intg points
FOLLOW_N	"Section follows nodes" flag	NEIPH	#"Extra" solid variables
		NEIPS	#"Extra" Shell variables
		NEIPT	#"Extra" Tk shell variables

Array subscript acronyms (for example a[X] is the same as a[0])

X	0 } directional	XX	0 }
Y	1 } equivalents	YY	1 } directional
Z	2 } for vectors	ZZ	2 } equivalents
		XY or YX	3 } for tensors
		YZ or ZY	4 }
		ZX or XZ	5 }

Data component acronyms and return value types:

All data components (as used by GetData()) return a single scalar value unless noted otherwise in the table below as **vector** or **tensor**, in which case the array data order is also given.

Nodal data component acronyms		**Solid and Shell specific data acronyms**	
BX	Basic (undeformed) X coord	VOL	Volume } solids
BY	Basic Y coord	RVOL	Relative volume } only
BZ	Basic Z coord		
BV	Basic **vector** [Bx,By,Bz]	THK	Thickness
		AREA	Area
CX	Current X coord		
CY	Current Y coord	RFX	FX force resultant
CZ	Current Z coord	RFY	FY ditto
CV	Current **vector** [Cx,Cy,Cz]	RFXY	FXY ditto
		RQX	XZ shear force ditto
DX	X displacement	RQY	YZ shear force ditto
DY	Y displacement	RMX	MX moment resultant
DZ	Z displacement	RMY	MY moment ditto
DM	Displacement magnitude	RMXY	MXY moment ditto
DV	Displacement **vector** [Dx,Dy,Dz]		
		EDEN	Internal energy density
VX	X velocity	HGEN	Hourglass energy
VY	Y velocity	TSTP	Timestep
VZ	Z velocity	MASS	Mass
VM	Velocity magnitude	MADD	Added mass
VV	Velocity **vector** [Vx,Vy,Vz]		
AX	X acceleration		
AY	Y acceleration		
AZ	Z acceleration		
AM	Acceleration magnitude		
AV	Acceleration **vector** [Ax,Ay,Az]		
TEMP	Nodal temperature		
Element stress components		Element strain components	

SXX	X stress	EXX	X strain
SYY	Y stress	EYY	Y strain
SZZ	Z stress	EZZ	Z strain
SXY or SYX	XY shear stress	EXY or EYX	XY shear strain
SYZ or SZY	YZ shear stress	EYZ or EZY	YZ shear strain
SZX or SXZ	ZX shear stress	EZX or EXZ	ZX shear strain
SVON	von Mises stress	EVON	von Mises strain
SMAX	Max principal stress	EMAX	Max principal strain
SMID	Middle princ stress	EMID	Middle princ strain
SMIN	Min princ stress	EMIN	Min princ strain
SAV	Average stress (pressure)	EAV	Average strain
SMS	Max shear stress	EMS	Max shear strain
STEN	Stress **tensor** [Sxx,Syy,Szz,Sxy,Syz,Szx]	ETEN	Strain **tensor** [Exx,Eyy,Ezz,Exy,Eyz,Ezx]
		EPL	Effective plastic strain
		ERATE	Strain rate
Basic and integrated beam components		**Belytschko-schwer resultant data**	
BFX	FX axial force	BSAX	Total axial strain
BFY	FY shear force	BPE1	Plastic energy at end 1
BFZ	FZ shear force	BPE2	Plastic energy at end 2
BFR	Force magnitude	BRY1	Y rotation end 1
		BRY2	Y rotation end 2
BMXX	MXX torsional moment	BRZ1	Z rotation end 1
BMYY	MYY bending moment	BRZ2	Z rotation end 2
BMZZ	MZZ bending moment	BMY1	Y moment end 1
BRM	Moment magnitude	BMY2	Y moment end 2
		BMZ1	Z moment end 1
BFMV	Force and moment **vectors** 6 values: [Fx,Fy,Fz,Mxx,Myy,Mzz]	BMZ2	Z moment end 2
		BAEN	Axial energy
		BIE	Internal energy
BSXX	Axial stress	BRXX	Torsional rotation
BSYX or BSXY	YZ shear stress		
BSZX or BSXZ	XZ shear stress	BBED	Bending energy density
BEP	Effective plastic strain	BAED	Axial energy density
BEAX	Axial strain	BIED	Internal energy density
"Extra" solid and shell data		**ALE data**	
SOX	Extra solid data	ADENS	Ale density
SHX	Extra shell & tk shell data	ADMOF	Ale dominant fraction
		AMMG	Ale multi-material group id
		AMMS	Ale multi-mat group mass
Contact surface derived data (only if a .ctf file has been read)			
CSN	Contact normal stress	**CFGX**	Contact global X force
CST	Contact tangential stress	**CFGY**	Contact global Y force
CSX	Contact local X stress	**CFGZ**	Contact global Z force
CSY	Contact local Y stress	**CFLX**	Contact local X force
		CFLY	Contact local Y force
		CFLZ	Contact local Z force
		CFM	Contact force magnitude
User-defined data component type codes			
UNOS	Node scalar	U_NODE	User-defined nodal component
UNOV	Node vector	U_SOSH	ditto solid, shell, tk shell comp
		U_BEAM	ditto beam components
USSS	Solid & shell scalar		
USST	Solid & shell tensor	U_SCALAR	Scalar data (1 value)
		U_VECTOR	Vector data (3 values)
UBMS	Beam scalar	U_TENSOR	Tensor data (6 values)
UBMV	Beam vector		

Examples

The following simple examples show how the functions above might be used. Further example scripts may be found in directory **$OASYS/d3plot_library/examples**

Capturing a sequence of static images to JPEG files

The following example loops over all frames in window #1 issuing the dialogue command "**/IMAGE JPG example_file_**nnn**.jpg**" which will capture each frame in a separate JPEG file.

```
n = GetWindowMaxFrame(1); // Here window #1 is assumed to be current

for (i=1; i<=n; i++)
{
    SetWindowFrame(1, i);
    DialogueInput("/IMAGE JPG example_file_" + i +".jpg");
}
```

Looping through states extracting cut-section results

The following example works through each state in the current model in turn, setting up a cut-section at the constant X coordinate of node #100, then rotates this section through 360 degrees in 5 degree increments, printing out the resulting forces and moments at each increment.

```
/* We need an array with 9 subscripts to hold data */

var data = new Array(9);

/* Turn on cut section display in window #1 */

SetCutSection(1, STATUS, ON);

/* Loop over all states in the model in turn */

n = GetNumberOf(STATE);

for(i=1; i<=n; i++)
{
    time = GetTime(i);
    Print("Time = " + time + "\n");

    SetCurrentState(i);

/* Set sections at const X at current position of node 100, and "get" the result
*/

    SetCutSection(1, CONST_X, 100);
    info = GetCutSection(1);

/* The following uses origin and vectors mode to sweep the section through 360
degrees in
** 5 degree increments, extracting the forces and moments at each position */

    data[0] = info.origin[X]; /* Origin stays where it currently is */
    data[1] = info.origin[Y];
    data[2] = info.origin[Z];

    for(i=0; i<=360; i+=5)
    {
        st = Math.sin(i*0.017453);
        ct = Math.cos(i*0.017453);

        data[3] = ct; /* X axis vector */
        data[4] = 0.0;
        data[5] = st;

        data[6] = 0.0; /* XY plane vector */
        data[7] = st;
        data[8] = ct;
```

```
        SetCutSection(1, OR_AND_V, data);

        c = GetCutForces(1);

        Print("Forces = " + c.force[X] + ", " + c.force[Y] + ", " + c.force[Z] +
"\n");
        Print("Moments = " + c.moment[X] + ", " + c.moment[Y] + ", " +
c.moment[Z] + "\n");
        Print("Centroid = " + c.centroid[X] + ", " + c.centroid[Y] + ", " +
c.centroid[Z] + "\n");
        Print("Area = " + c.area + "\n\n");
    }
}
```

Calculating the max value and storing it as a UBIN component.

This example loops over all states in model #1 finding the maximum Sxx value of each solid and shell in the model, and storing it as a new scalar UBIN component in state #1.

```
/* If you have > 1 model then set the one you want. Model #1 is assumed in
** this example. */

SetCurrentModel(1);

/* It is assumed that only a single scalar value is required, and it is for
** solids & shells. So find out the number of states, solids and shells */

nstate = GetNumberOf(STATE);

nshell = GetNumberOf(SHELL);
nsolid = GetNumberOf(SOLID);

shell_env = new Array();
solid_env = new Array();

/* Create arrays to hold the max/min data. Here they are initialised to
** zero, which might not be a good choice if you are looking for max values
** and incoming results could be negative. Fill in your own initial value. */

for(j=1; j<=nshell; j++) shell_env[j] = 0.0;
for(j=1; j<=nsolid; j++) solid_env[j] = 0.0;


/* Loop over states collecting max data.
**
** Note that making the outer loop the state is more efficient than making it
the
** element, since changing state is a more costly operation. */

for(i=1; i<=nstate; i++)
{
    SetCurrentState(i);
    Print("Doing state " + i + "\n");

    for(j=1; j<=nshell; j++)
    {
        c = GetData(SXX, SHELL, j, 1);
        if(c > shell_env[j]) shell_env[j] = c;
    }

    for(j=1; j<=nsolid; j++)
    {
        c = GetData(SXX, SOLID, j, 1);
        if(c > solid_env[j]) solid_env[j] = c;
    }
}

/* Now create a scalar user-defined binary component to hold the result */
```

```
icomp = CreateUbinComponent("Maximum of SXX", U_SOSH, U_SCALAR);

/* Data has to be stored at a state, so choose state 1 */

SetCurrentState(1);

/* Then populate this user-defined component, intg point #1. */

for(j=1; j<=nshell; j++) PutUbinData(icomp, SHELL, j, 1, shell_env[j]);
for(j=1; j<=nsolid; j++) PutUbinData(icomp, SOLID, j, 1, solid_env[j]);
```

Extracting data by ply from a composite analysis.

Post-processing composite analyses can be difficult because the elements have many integration points, and a physical ply may not use the same integration point in two adjacent elements, meaning that post-processing by "layer" (ie by integration point) is not helpful.

This script assumes that ply information has been created using a *PART_COMPOSITE card, with each ply assigned to a separate material id of the relevant ply number, and it sorts results into a separate UBIN component by ply. In this way a given UBIN component will show results for a single ply across the whole model.

```
// JavaScript to plot composite data for selected elements
// Date: 26 June 2008
// Version 1.0

n = get_window_max_frame(1);

set_current_model(1);
var a = GetWindowFrame(1);
SetCurrentState(a);

nsh = GetNumberOf(SHELL);
nip = GetNumberOf(NIP_S);
npa = GetNumberOf(PART);

var part = new Array();
var part_id = new Array();
var part_flag = 0;
var co1 = 0; // part counter
var co2 = 0; // ply counter

var ply = new Array();
var ply_id = new Array();
var ply_flag = 0;

var tmp1 = 0;

var offset = {}; // part, ip
var order1= {}; // part, ip
var order2= {}; // part, ip

var max1 = {};
var max2 = {};

var iter= 0; // iteration counter
var i_flag = 0;
var t_flag = 0;


// setup part array

for (i=1; i<=npa; i++) //shell
{
    part_flag = 0;
    a = GetLabel(PART, i);
    b = GetMid(PART, i, 1);

    if (b == 0) // check to see if the part is not a Part Composite
    {
```

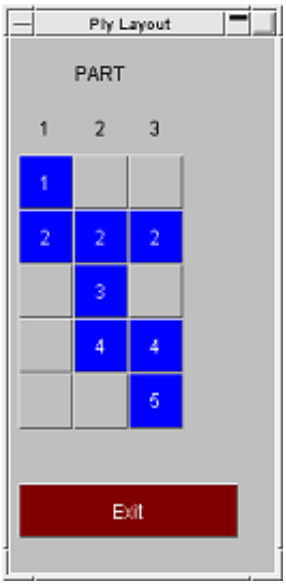
```
        part_flag = 1;
    }

    for(j=1; j<=co1; j++) //part
    {
        if (a == part_id[j]) // check to see if the part is already defined
        {
            part_flag = 1;
        }
    }

    if (part_flag == 0)
    {
        co1 = co1 + 1;
        part[co1] = i;
        part_id[co1] = a;
    }
}

// setup ply order array

for (i=1; i<=co1; i++) // part
{
    order1[i] = new Object;

    for(j=1; j<=nip; j++) // ip
    {
        b = GetMid(PART, part[i], j);
        order1[i][j] = b;
    }
}

// setup ply id array

for (i=1; i<=co1; i++) //part
{
    for(j=1; j<=nip; j++) //ip
    {
        b = order1[i][j];

        if (b == 0) // ply doesn't exist
        {
            ply_flag = 1;
        }
        for (k=1; k<=co2; k++) //ply already created
        {
            if (b == ply_id[k])
            {
                ply_flag = 1;
            }
        }

        if (ply_flag == 0)
        {
            co2 = co2 + 1;
            ply_id[co2] = b;
            Message(co2 +" "+ b);
        }

        ply_flag = 0;
    }
}

//////////////////////////// proccessing loop

while (i_flag == 0)
{

    iter = iter + 1;
    Message("Iteration "+ iter);
```

```
    if(iter > 1)
    {
        for (i=1; i<=co1; i++) // part
        {
            for(j=1; j<=co2; j++) // ip
            {
                order1[i][j] = order2[i][j];
            }
        }
    }

//// ply order array

    for (i=1; i<=co2; i++) //ply
    {
        ply[i] = new Object;

        for(j=1; j<=co1; j++) //part
        {
            ply[i][j] = new Object;

            for(k=1; k<=co2; k++) //ply
            {
                b = order1[j][k];
                if (b == ply_id[i])
                {
                    ply[i][j] = k
                }
            }
        }
    }

//// find max position for each ply

    for (i=1; i<=co2; i++)
    {
        max1[i] = 0;

        for(j=1; j<=co1; j++)
        {
            if (typeof(ply[i][j]) == "object")
            {
                ply[i][j] = 0;
            }
            max1[i] = Math.max(max1[i], ply[i][j] );
        }
    }

    for (i=1; i<=co2; i++)
    {
        Message(i +" "+ ply_id[i] + " "+ max1[i]);
    }

// offset ply

    for (i=1; i<=co1; i++) // part
    {
        order2[i] = new Object;
        max = 0;
        pos2 = 0;

        for(j=1; j<=co2; j++) // ip
        {
            pos1 = j;
            b = order1[i][j]; // ply id
            for(k=1; k<=co2; k++) // ply
            {
                if(ply_id[k] == b)
                {
                    max = max1[k]
                }
```

```
            }

            if(b != null && b != 0)
            {
                p_off = Math.max((pos2 - pos1 + 1),0);
                pos2 = Math.max(max, (p_off + pos1));
                order2[i][pos2] = b;
            }
        }
    }

    i_flag = 1;

    for (i=1; i<=co1; i++) // part
    {
        for(j=1; j<=co2; j++) // ip
        {
            if(order1[i][j] != order2[i][j]) i_flag = 0;
        }
    }
}
////////////////// create table


var we = new Window("Ply Layout", 0.2, 0.3, 0.5, 0.6 );
var x1 = 0;
var x2 = 0;
var y1 = 0
var y2 = 0;

var part = new Widget(we, Widget.LABEL, 0, Math.max((co1*10)+30, 70), 10, 20,
"PART");
part.justify=Widget.CENTRE;

for (i=1; i<=co2; i++) // ply
{
    t_flag = 1;
    y1 = (i*10) + 20;
    y2 = y1 + 10;

    for(j=1; j<=co1; j++) // part
    {
        x1 = (j*10) + 10;
        x2 = x1 + 10;

        if(i == 1)
        {
            var partl = new Widget(we, Widget.LABEL, x1, x2, 20, 30,
""+part_id[j]);
        }

        var mark = new Widget(we, Widget.BUTTON, x1, x2, y1, y2, " ");
        if (order2[j][i] != 0 && order2[j][i] != null)
        {
            mark.text= ""+order2[j][i];
            mark.background=Widget.BLUE;
            mark.foreground=Widget.WHITE;
            t_flag = 0;
        }
    }
    if(t_flag == 1)
    {
        var yn = i;
        i = co2+1;
    }
}

var ply = new Widget(we, Widget.LABEL, 10, 20, (yn*5)+25, (yn*5)+35, "PLY");

var exit = new Widget(we, Widget.BUTTON, 20, 60, (y1+20), (y2+20), "Exit");
exit.background = Widget.DARKRED;
```

```
exit.foreground = Widget.WHITE;
exit.onClick = ex_clicked;

we.Show(false)

////////////////////////////////////////////////////////////////////////////
function ex_clicked()
{
    Exit();
}
////////////////////////////////////////////////////////////////////////////
```

Using the Window and Widget classes to present ply data graphically



This example demonstrates the use of Menu Window and Widget classes of the common API shared with PRIMER to present information to the user graphically, and to interact with him.

It goes through a laminate model using *PART_COMPOSITE and sorts the plies into usage by part. This information is then presented to the user as shown in the figure here and the script pauses until the user clicks on "Exit" in order to continue.

```
SetCurrentModel(1);
SetCurrentState(1);

nsh = GetNumberOf(SHELL);
nip = GetNumberOf(NIP_S);
npa = GetNumberOf(PART);

var part = new Array();
var part_id = new Array();
var part_flag = 0;
var co1 = 0; // part counter
var co2 = 0; // ply counter

var ply = new Array();
var ply_id = new Array();
var ply_flag = 0;

var tmp1 = 0;

var offset = {}; // part, ip
var order1= {}; // part, ip
var order2= {}; // part, ip

var max1 = {};
var max2 = {};

var iter= 0; // iteration counter
var i_flag = 0;
var t_flag = 0;


// setup part array

for (i=1; i<=nsh; i++) //shell
{
    part_flag = 0;
```

```
    a = GetPid(SHELL, i);
    a = GetLabel(PART, a);
    b = GetMid(SHELL, i, 1);

    for(j=1; j<=co1; j++) //part
    {
        if (a == part_id[j] || b == 0) // check to see if the part is already
defined or not a Part Composite
        {
            part_flag = 1;
        }
    }

    if (part_flag == 0)
    {
        co1 = co1 + 1;
        part[co1] = i;
        part_id[co1] = a;
    }

}

// setup ply order array

for (i=1; i<=co1; i++) // part
{
    order1[i] = new Object;
    for(j=1; j<=nip; j++) // ip
    {
        b = GetMid(SHELL, part[i], j);
        order1[i][j] = b;
    }
}

// setup ply id array

for (i=1; i<=co1; i++) //part
{
    for(j=1; j<=nip; j++) //ip
    {
        b = order1[i][j];
        for (k=1; k<=co2; k++) //ply
        {
            if (b == ply_id[k] || b == 0)
            {
                ply_flag = 1
            }
        }
        if (ply_flag == 0)
        {
            co2 = co2 + 1;
            ply_id[co2] = b;
        }
        ply_flag = 0;
    }
}

//////////////////////////// processing loop

while (i_flag == 0)
{
    iter = iter + 1;
    Message("Iteration "+ iter);

    if(iter > 1)
    {
        for (i=1; i<=co1; i++) // part
        {
            for(j=1; j<=co2; j++) // ip
            {
                order1[i][j] = order2[i][j];
```

```
            }
        }
    }

//// ply order array

    for (i=1; i<=co2; i++) //ply
    {
        ply[i] = new Object;
        for(j=1; j<=co1; j++) //part
        {
            ply[i][j] = new Object;
            for(k=1; k<=co2; k++) //ply
            {
                b = order1[j][k];
                if (b == ply_id[i])
                {
                    ply[i][j] = k
                }
            }
        }
    }

//// find max position for each ply

    for (i=1; i<=co2; i++)
    {
        max1[i] = 0;
        for(j=1; j<=co1; j++)
        {
            if (typeof(ply[i][j]) == "object")
            {
                ply[i][j] = 0;
            }
            max1[i] = Math.max(max1[i], ply[i][j] );
        }
    }


// offset ply

    for (i=1; i<=co1; i++) // part
    {
        tmp1 = 0;
        offset[i] = new Object;
        for(j=1; j<=co2; j++) // ip
        {
            b = order1[i][j];
            for(k=1; k<=co2; k++) // ply
            {
                if(ply_id[k] == b)
                {
                    if(max1[k] > j)
                    {
                        tmp1 = max1[k] - j;
                    }
                }
            }
            offset[i][j] =tmp1
        }
    }

    for (i=1; i<=co1; i++) // part
    {
        order2[i] = new Object;
        for(j=1; j<=co2; j++) // ip
        {
            a = offset[i][j];
            b = order1[i][j];
            order2[i][j+a] = b;
        }
```

```
    }

    i_flag = 1;

    for (i=1; i<=co1; i++) // part
    {
        for(j=1; j<=co2; j++) // ip
        {
            if(offset[i][j] != 0) i_flag = 0;
        }
    }
}

////////////////// create table


var we = new Window("Ply Layout", 0.2, 0.3, 0.5, 0.6 );
var x1 = 0;
var x2 = 0;
var y1 = 0
var y2 = 0;

var part = new Widget(we, Widget.LABEL, (co1*5)+15, (co1*5)+25, 10, 20, "PART");
part.justify=Widget.CENTRE;

for (i=1; i<=co2; i++) // ply
{
    t_flag = 1;
    y1 = (i*10) + 20;
    y2 = y1 + 10;

    for(j=1; j<=co1; j++) // part
    {
        x1 = (j*10) + 10;
        x2 = x1 + 10;

        if(i == 1)
        {
            Message("a"+ part_id[j]);
            var partl = new Widget(we, Widget.LABEL, x1, x2, 20, 30,
""+part_id[j]);
        }

        var mark = new Widget(we, Widget.BUTTON, x1, x2, y1, y2, " ");
        if (order2[j][i] != 0 && order2[j][i] != null)
        {
            mark.text= ""+order2[j][i];
            mark.background=Widget.BLUE;
            mark.foreground=Widget.WHITE;
            t_flag = 0;
        }
    }
    if(t_flag == 1)
    {
        var yn = i;
        i = co2+1;
    }
}

var ply = new Widget(we, Widget.LABEL, 10, 20, (yn*5)+25, (yn*5)+35, "PLY");

var exit = new Widget(we, Widget.BUTTON, 20, 60, (y1+20), (y2+20), "Exit");
exit.background = Widget.DARKRED;
exit.foreground = Widget.WHITE;
exit.onClick = ex_clicked;

we.Show(false)

////////////////////////////////////////////////////////////////////////////
function ex_clicked()
{
```

```
    Exit();
}
```

Using the File class to write data to file.

This example extracts maximum and minimum Z displacement values of nodes and writes them to a file "**reporter_variables**" that can be used to set up variables in Oasys Reporter. It demonstrates the use of the File class to open a file, write to it, and close it.

```
var fence_part_min = 4;
var fence_part_max = 7;

var max_states;
var istate;
var ipart;
var nnode;
var inode;
var objElem;
var internal_pid;
var external_pid;

var z_max = 0;
var z_min = 0;
var max_node = 0;
var min_node = 0;
var max_state = 0;
var min_state = 0;

var f;

/* Get the number of states in the current model */

max_states = GetNumberOf(STATE);

/* Get the number of nodes in the current model */

nnode = GetNumberOf(NODE);

print("Start of loop\n");

/* Loop over each state */

for(istate=1; istate<=max_states; istate++)
{
    SetCurrentState(istate);

    print("State " + istate + "\n");

/* Loop over each node */

    for(inode=1; inode<=nnode; inode++)
    {

/* Get shell elements at node */

        if(objElem = GetElemsAtNode(inode, SHELL))
        {
            if(objElem.nn > 0)
            {

/* Get the external PID */

                internal_pid = GetPid(SHELL, objElem.list[0]);
                external_pid = GetLabel(PART, internal_pid);

/* Check it against parts to test */

                if(external_pid>=fence_part_min && external_pid<=fence_part_max)
                {
                    temp = GetData(DZ, NODE, inode);
```

```
                    if(temp > z_max)
                    {
                        max_state = istate; /* Store the state the maximum
occurs */
                        max_node = inode; /* Store the node the maximum occurs
at */
                        z_max = temp; /* Store the maximum */
                    }
                    if(temp < z_min)
                    {
                        min_state = istate; /* Store the state the minimum
occurs */
                        min_node = inode; /* Store the node the minimum occurs
at */
                        z_min = temp; /* Store the minimum */
                    }
                }
            }
        }
    }
}

/* Open a file to write the variables to */

f = new File("./reporter_variables", File.WRITE);

/* Write to the file */

f.Writeln("VAR Z_MAX DESCRIPTION='Maximum z displacement' VALUE='" + z_max +
"'");
f.Writeln("VAR Z_MIN DESCRIPTION='Minimum z displacement' VALUE='" + z_min +
"'");
f.Writeln("VAR Z_MAX_NODE DESCRIPTION='Node with maximum z displacement'
VALUE='" + max_node + "'");
f.Writeln("VAR Z_MIN_NODE DESCRIPTION='Node with minimum z displacement'
VALUE='" + min_node + "'");
f.Writeln("VAR Z_MAX_STATE DESCRIPTION='State with maximum z displacement'
VALUE='" + max_state + "'");
f.Writeln("VAR Z_MIN_STATE DESCRIPTION='State with minimum z displacement'
VALUE='" + min_state + "'");

/* Close the file */

f.Close();
```

Next section